Watershed Modeling System

WMS v8.1

TUTORIALS – Volume 2

WATERSHED MODELING

WMS 8.1 Tutorials – Volume 2

The software *WMS* is a product of the Aquaveo, LLC. For more information about this software and related products, contact Aquaveo at:

Aquaveo
75 South 200 East, Suite 201
Provo, Utah 84606
Tel.: (801) 691-5528
e-mail: wms@aquaveo.com
WWW: http://www.aquaveo.com/

For technical support, contact Aquaveo's tech support number at (801) 691-5530 (Monday-Friday, 8am-5pm Mountain Time)

TABLE OF CONTENTS

CHAPTER 1

DEM Delineation

Watershed delineation from DEMs is straightforward and relatively simple, provided the project area is not entirely flat or completely dominated by man-made structures (you cannot expect the DEM method to work if there is no relief in the DEM elevations themselves). This exercise teaches DEM delineation using the hydrologic modeling wizard, a step-by-step delineation approach that makes the process even simpler.

1.1 Objectives

In this exercise you will learn the basics of DEM delineation using the hydrologic modeling wizard. This includes the following:

1. Importing DEM Data
2. Computing flow paths and flow accumulations
3. Delineating watersheds from DEMs
4. Delineating sub-basins within a watershed

1.2 Importing DEM Data

The first step in delineating a watershed is to import one or more DEMs. Though DEM retrieval will be automated through the hydrologic modeling wizard at some point in the future, for this exercise we will import the DEM data manually.

To read in a set of four 30-meter DEMs from the 1:24000 series, complete the following steps:

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the folder ***C:\Program Files\WMS81\tutorial\demdelin***
5. Open *"josephpeak.dem"*
6. In the Importing USGS DEMs dialog, click the *Add* button
7. Add the following three DEMs (remember that you can hold the CTRL key down and add more than one at once, or use the *Add* button to add each additional one before selecting *OK*):
    - *marysvalecanyon.dem*
    - *redridge.dem*
    - *trailmountain.dem*
8. Enter a thinning factor of **3** in the Thinning factor edit field

Thinning the resolution of the DEMs will reduce the density of elevation points so that your computer will process the DEM data faster. The resolution of points in the 30-meter DEMs is too dense for the purposes of this exercise, so you will not lose any accuracy by thinning.

9. Select *OK*

1.2.1 Trimming the DEM

1. Select ***DEM | Trim | Polygon***
2. Select the *Enter a polygon interactively* option
3. Select *OK*
4. Trace around the area formed by the polygon shown in Figure 1-1 clicking on each corner and double-clicking to end
5. Select the *Frame* macro

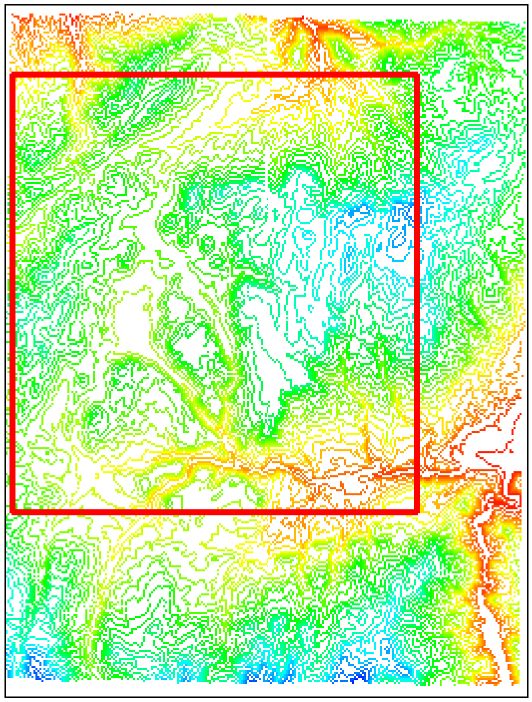


Figure 1-1: DEM Trim Area

6. Right-click on *DEM* under *Terrain Data* in the Project Explorer and select ***Fill*** to linearly interpolate and fill gaps that were created by thinning the DEM data

1.3 Watershed Delineation using the Hydrologic Modeling Wizard

Now that we've imported a DEM, we can launch the Hydrologic Modeling Wizard to guide us through the process of watershed delineation.

1. Locate the icon for the *Hydrologic Modeling Wizard* at the bottom of your WMS window. Click the icon to open the wizard 

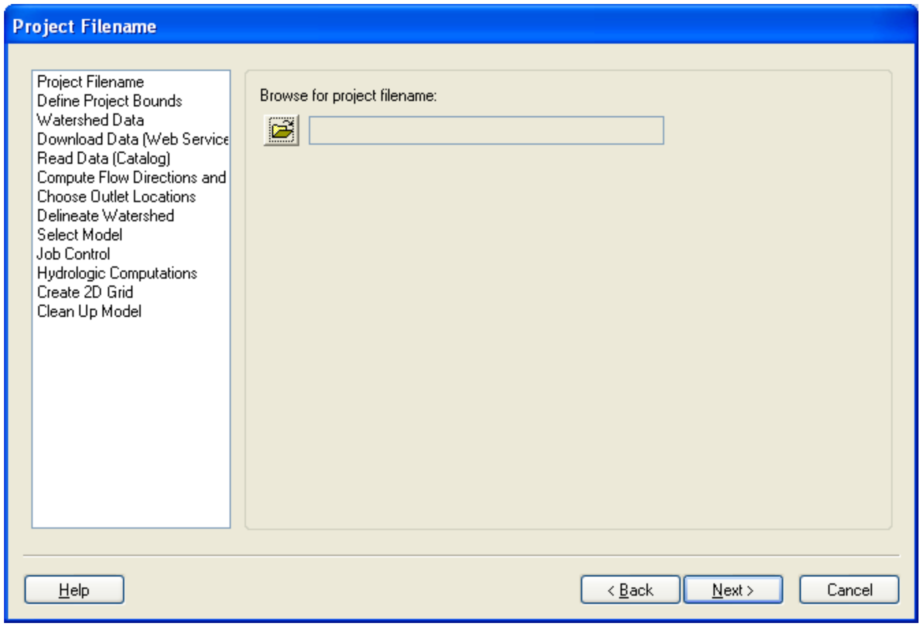


Figure 1-2: Hydrologic Modeling Wizard

The Hydrologic Modeling Wizard window shown in Figure 1-2 consists of two sections. The list box on the left shows the steps needed to set up your hydrologic model. The area on the right displays specific tasks associated with the selected step. Clicking Next > on the wizard window will walk you through the steps of the wizard sequentially. However, you may skip to any step in the wizard at any time by clicking on the associated heading to the left.

1.3.1 Project Filename

1. Make sure *Project Filename* is the current window in the Hydrologic Modeling Wizard
2. Select the file browser button to browse for a project filename 
3. Name the project "*DemDelineation.wpr*" and click *Save*
4. Click *Next >* to advance to the next step

1.3.2 Define Project Bounds

We will leave the project coordinate system the same as that associated with the DEMs we imported earlier.

1. Under Project coordinate system, select *Define…*
2. Select *Global Projection*

3. Select *Set Projection*

4. Set the *Projection* to *UTM*, *Datum* to *NAD 27*, *Planar Units* to *METERS*, and *Zone* to *12 (114°W - 108°W – Northern Hemisphere)*

5. Select *OK*

6. Set Vertical Units to *Meters*

7. Select *OK*

8. Under Project Boundary, select *Define…*

9. In the Microsoft Virtual Earth Map Locator window, enter a latitude of **38.6469** and a longitude of **-112.4291** and click jump to location

10. Zoom in until the Virtual Earth window looks similar to Figure 1-3 and then click *OK*

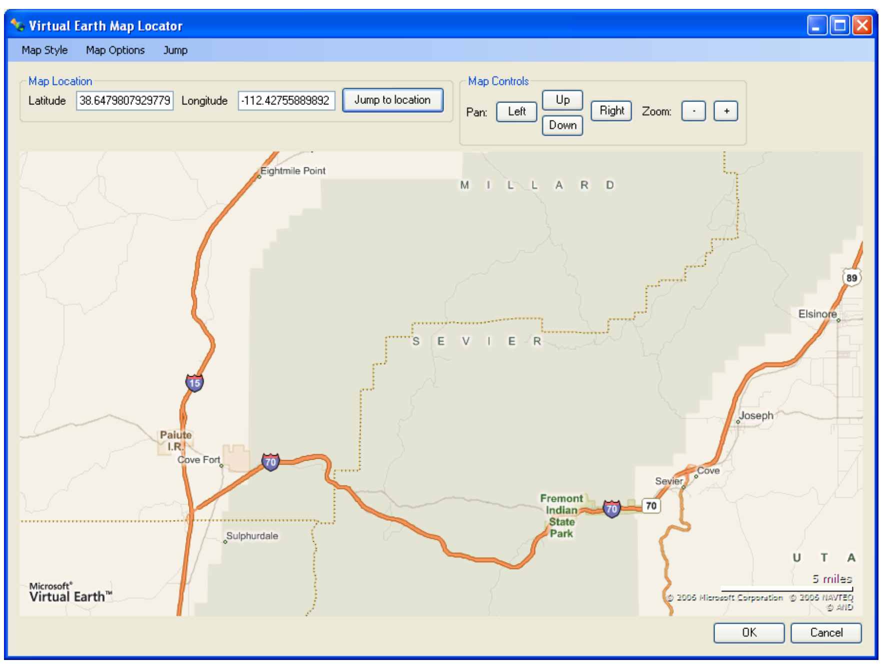


Figure 1-3: Project Bounds in Virtual Earth

11. Click *Next* > to advance to the next step

1.3.3 Download Watershed Data from Web Services

1. Select *Web Services* as the data source

2. Click *Next* > to advance to the next step

3. In the Data Type column of the Web Services spreadsheet, toggle on the option for *TerraServer aerial photo*

4. Click the *Browse…* button next to TerraServer aerial photo

5. Name the file "*wizphoto.jpg*" and click Save

6. In the Data Type column of the Web Services spreadsheet, toggle on the option for *TerraServer topo*

7. Click the *Browse…* button next to TerraServer topo

8. Name the file "*wiztopo.jpg*" and click Save

9. Make sure the options for *NED data*, *SRTM data*, and *TerraServer urban* are toggled off, then click *Download Data from Web*

WMS will proceed to download the selected files for the project area you specified.

10. Click *OK* to use the recommended map scale

11. Select *Yes* if asked to generate pyramids

12. Repeat steps 10 and 11 for the second image

Once the image files have been downloaded from Web Services, WMS automatically opens the files into your WMS project.

13. Click *Next >* to advance to the next step

1.3.4 Computing Flow Directions and Accumulations

WMS computes flow directions and flow accumulations to create streams on the DEM using a program called TOPAZ.

1. Make sure the option to *Write TOPAZ files to a temp directory* is selected and that the selected sub-basin areas are in *Square Miles* and that distances are computed in *Feet*

2. Select *Compute TOPAZ*

3. Click Close when TOPAZ terminates

4. Click *Next >* to advance to the next step

1.3.5 Define Outlets and Delineate Basins

The Hydrologic Modeling Wizard window is a non-modal window, meaning it lets you interact with the main model window with the window still open

1. Zoom in around the area shown in Figure 1-4

2. Choose the Create Outlet Point tool from the wizard window 

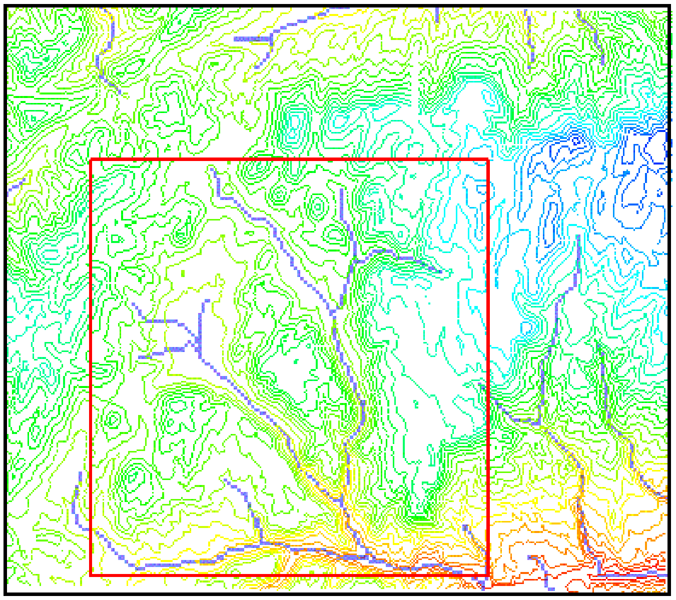


Figure 1-4: DEM Zoom Area

3. Place the outlet just upstream of the branch junction shown in Figure 1-5

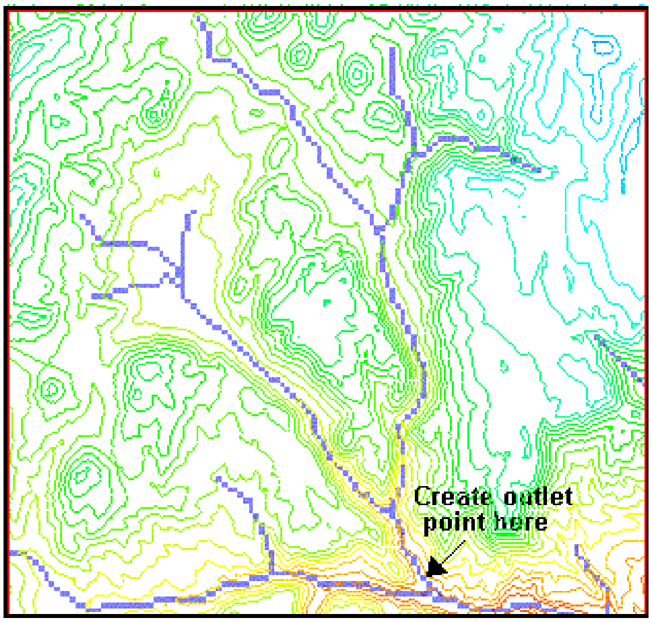


Figure 1-5: Drainage Outlet Location

4. Click *Next* > to advance to the next step

5. Enter a stream threshold value of 1.00 mi^2 and the Computation Units at their default values

6. Select *Delineate Watershed*

When you select Delineate Watershed, WMS converts the DEM streams to feature arcs, defines the basin boundary, and computes basin related data including basin area, average basin slope, mean basin elevation, and time of concentration.

7. Click *Next* > to advance to the next step

1.3.6 Create Sub-basins

Since we won't be setting up a hydrologic model in this exercise, we don't need to complete the rest of the steps in the wizard. However, we would like to go back and create sub-basins.

1. In the list box on the left-side of the wizard window, select the heading *Choose Outlet Locations*

2. Zoom in around the area shown in Figure 1-6

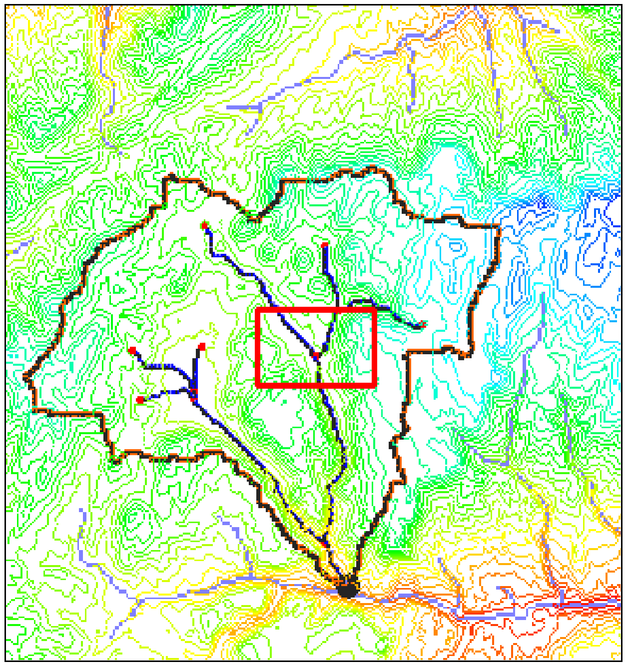


Figure 1-6: Zoom Area

3. Choose the Create Outlet Point tool from the wizard window

4. Create another outlet at the location shown in Figure 1-7, just below the stream junction.

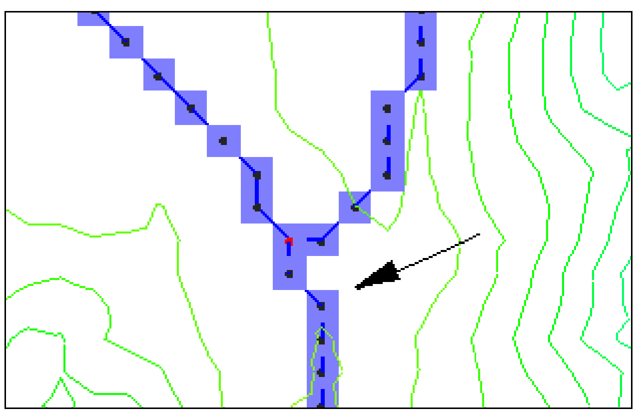


Figure 1-7: Node Location

5. Choose the *Frame* macro

6. Zoom in around the branch shown in Figure 1-8

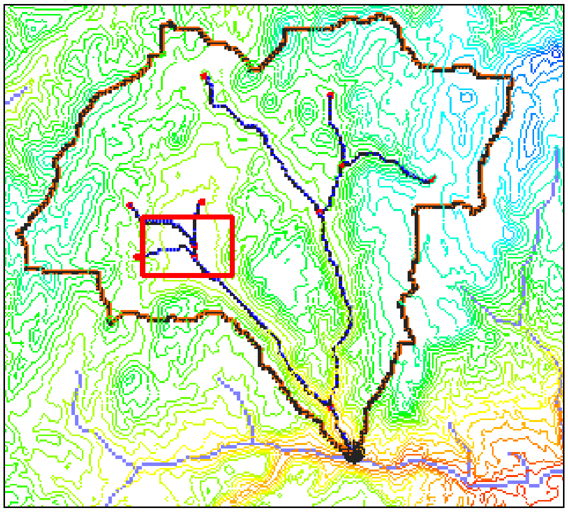


Figure 1-8: Zoom Area

7. Choose the Create Outlet Point tool from the wizard window 

8. Place an outlet immediately downstream from the most downstream node you can see, as shown in Figure 1-9

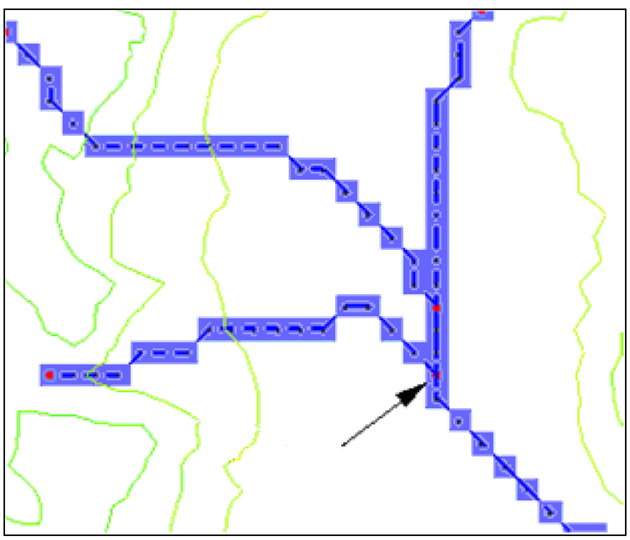


Figure 1-9: Change the indicated node to an outlet.

9. *Click* Next > *to advance to the next step*

10. Select *Delineate Watershed*

11. Click *OK* to delete the old basin boundary and create a new one

12. Click Cancel to close the Hydrologic Modeling Wizard

1.4 Displaying DEMs

WMS has several options for displaying DEMs and the results of basin delineation from a DEM. In this section you will explore a few of these options.

1.4.1 Move Basin Labels

You may want to move the basin labels displaying the area, slope, etc. to more convenient locations on the image.

1. Make sure the *Drainage* module is selected and Choose the *Move Basin Label* tool
2. Click on any area in the basin you just selected and drag the mouse to a location outside the basin before letting go of the mouse button

An arrow will be drawn from where you let go of the mouse to where you started.

1.4.2 Turning Displays Off

When you are finished using the elevation data, you may want to turn the DEM contours and extra streams displays off.

1. Right-click on *DEM* in the Project Explorer and select ***Display Options***
2. Uncheck the *Flow Accumulation*, *DEM Contours*, *Color Fill Drainage Basins*, and *Fill Basin Boundary Only* options
3. Select *OK*

1.4.3 Color Filling Basins

WMS also allows you to fill in each basin with a different color. This is useful when the background image does not need to be showing.

1. Right-click on the *Drainage* coverage and select *Zoom to Layer*
2. Right-click on the *Drainage* coverage and select ***Display Options***
3. Check the *Color Fill Polygons* option
4. Select *OK*

1.4.4 Smoothing Boundaries

If you zoom in near the boundary of one of your basins, you will see that the boundary lines are not smooth because they are formed by tracing the DEM cells. WMS allows you to redistribute vertices to smooth these boundaries.

1. Switch to the *Map* module
2. Choose the *Select Feature Arc* tool
3. Select ***Edit | Select All***
4. Select ***Feature Objects | Redistribute***
5. Select the *Use Cubic Spline* option
6. Enter **100** in the Spacing edit box
7. Select *OK*

When you zoom in on the basin boundaries now, they should be much smoother.

1.4.5 Color Fill Contours

WMS has several options for contouring DEMs.

1. Select ***Display | Display Options***
2. Toggle off the *Color Fill Polygons* option
3. In *DEM Data*, toggle on *DEM Contours*
4. Click the *Options* button next to *DEM Contours* and change the *Contour Method* to *Color Fill* and click *OK*

1.4.6 Saving Your Work

You can now resave your work as a WMS project.

1. Select ***File | Save As***
2. Save the file as *"DemDelineation.wpr"*
3. Select *Save* and *OK* to overwrite the existing project file

1.5 More Basin Delineation

In this exercise you have learned the most basic use of WMS: to delineate a watershed from a DEM. It would be a good idea to practice again using a DEM of your own choosing. In a previous chapter you were instructed to download your own DEM from the National Elevation Data Set site; it would be good here to gain further practice by delineating a watershed using this DEM. If you did not download a DEM, or would like to try a different area go to the GSDA website at http://xmswiki.com/wiki/GSDA:GSDA and follow the DEM links to download a DEM (be sure to convert the format to GridFloat from ArcGrid before downloading).

Follow these steps using your own data (using the previous sections for a more specific outline):

1. Download your DEM data in the GridFloat format and unzip
2. Start WMS, or choose the ***File | New*** option if continuing
3. Load the DEM and convert from *Geographic Coordinates* to *UTM NAD 83*
4. Crop out a portion of the DEM if it is much larger than the watershed you want to delineate (you might want to wait and do this after the next two steps if you are unsure)
5. Step through the Hydrologic Modeling Wizard and try locating the aerial and topo images that correspond to the DEM you downloaded
6. Delineate a basin and sub-basins if you'd like by stepping through the rest of the wizard
7. Experiment with some of the display settings to get a nice final map
8. Save a project file of your work

1.6 Conclusion

In this exercise you should have learned how to use DEM data in WMS. This includes the following:

1. Importing DEM Data
2. Computing flow paths and flow accumulations
3. Delineating watersheds from DEMs
4. Delineating sub-basins within a watershed

CHAPTER 2

Editing DEMs

Some terrain features, including man-made features such as roads, canals, reservoirs, dams, dikes, and levees, are not well represented in DEMs, especially if the DEM resolution is coarse. In addition, it may be desirable to evaluate future alterations in terrain that result from development scenarios. WMS has tools for editing DEM elevations and flow directions in order to include these features in the surface representation.

2.1 Objectives

In this exercise you will learn how to edit DEMs for more accurate surface representation and drainage analysis by completing the following steps:

1. Filling gaps of data with the DEM Fill command
2. Using stream arcs to edit flow directions and modify delineations
3. Editing flow directions of individual DEM points

2.2 DEM Fill Command

The DEM Fill command is useful for filling gaps in DEM data. It interpolates missing values using inverse distance weighting of the neighboring eight cells. After delineating an erroneous basin we will fill gaps in the data with the DEM Fill command, recompute flow accumulations and directions, and delineate the correct basin.

2.2.1 Open DEMs

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the folder ***C:\Program Files\WMS81\tutorial\demedit***
5. Open *"mvcanyon.dem"* and *"trailmount.dem"*
6. Select *OK* on the Importing USGS DEMs dialog

2.2.2 Run TOPAZ

1. Switch to the *Drainage* module
2. Select ***DEM | Compute TOPAZ Flow Data***
3. Select *OK*
4. Select *OK*
5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
6. Right-click on *DEM* in the Project Explorer and select ***Display Options***
7. Change the Minimum Accumulation For Display to **0.06** mi^2
8. Select *OK*
9. Zoom in to the top middle section of the DEM as shown in Figure 2-1

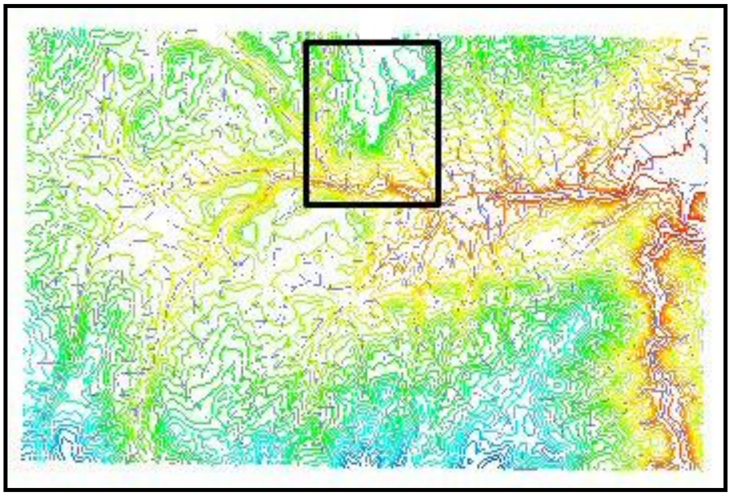


Figure 2-1: Zoom in to the rectangle on the DEM

2.2.3 Basin Delineation

1. Switch to the *Drainage* module 
2. Select the *Create Outlet Point* tool 
3. Click anywhere on the DEM to create an outlet
4. Select *OK* if you get a message telling you that the outlet is not located in a flow accumulation cell
5. Enter a Feature Point X-value of **379589.5** and a Feature Point Y-value of **4271008.5** in the Properties window on the right of the screen to edit the outlet location
6. Select ***DEM | Delineate Basins Wizard***
7. Select *OK*
8. Select *OK*

2.2.4 Basin Delineation Errors

The delineation of this basin, shown in Figure 2-2, looks suspicious. First of all, the upper right edge of the basin boundary is linear and flat. This usually indicates that the basin actually continues farther, but was not properly delineated for some reason. The other noticeable item is the gap in the flow accumulation cells. It looks like the stream disappears for a while. This is indicative of gaps in the DEM data.

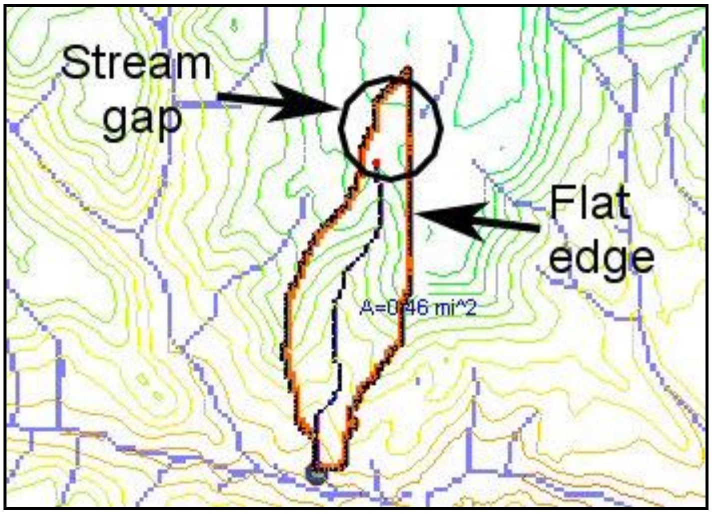


Figure 2-2: Delineation errors

1. Right-click on *DEM* and select ***Display Options***
2. On the DEM tab toggle *No Data Cells* on
3. Select *OK*

No Data cells that interfere with the basin delineation as shown in Figure 2-3.

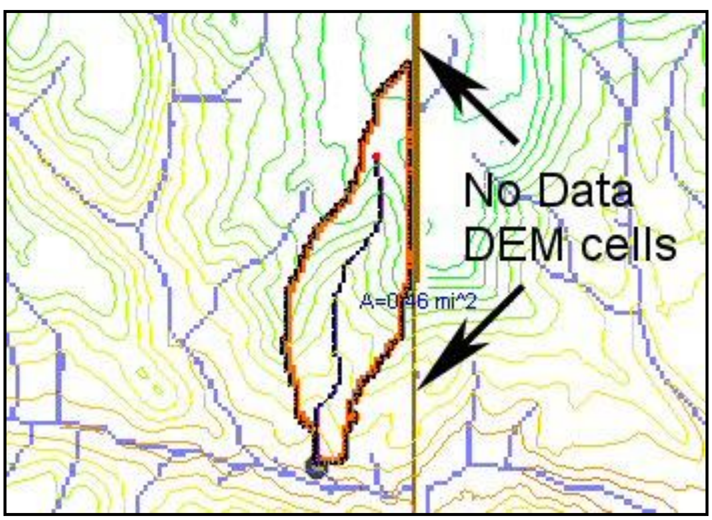


Figure 2-3: No Data cells cause incorrect basin delineation

2.2.5 Fill Data Gaps

1. Select ***Display | Frame Image***
2. Right-click on the DEM in the Project Explorer and select ***Fill***

3. Select *OK* if an information message appears

Notice that values were interpolated for the No Data cells lying in the interior part of the DEM.

2.2.6 Run TOPAZ

1. Switch to the *Drainage* module
2. Select ***DEM | Compute TOPAZ Flow Data***
3. Select *OK*
4. Select *OK*
5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
6. Use the *Zoom* tool to zoom back in around the basin

The flow accumulation cells now connect all the way through as shown in Figure 2-4.

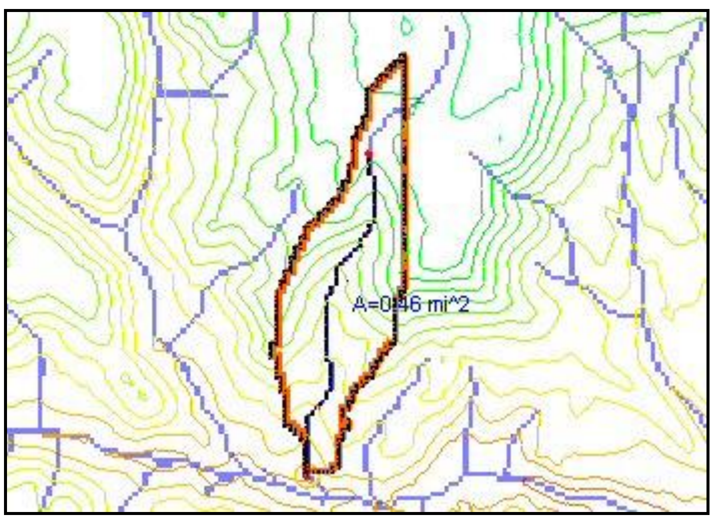


Figure 2-4: New TOPAZ results after using the DEM | Fill command

2.2.7 Basin Delineation

1. Select ***DEM | Delineate Basins Wizard***
2. If prompted, select *OK* to delete all existing feature data and recreate using the new basin delineation

3. Select *OK*

4. Select *OK*

Figure 2-5 shows the correct basin delineation.

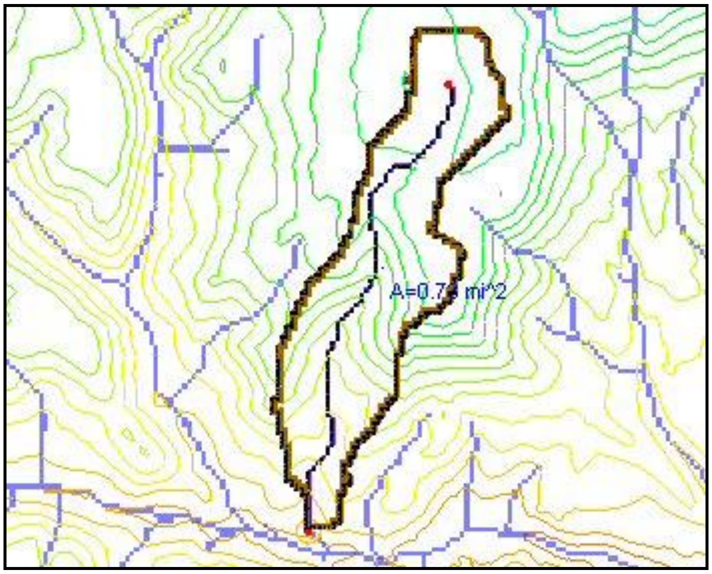


Figure 2-5: Final basin delineation

2.3 Adding Stream Arcs to Edit Flow Directions and Basins

Sometimes you will need to add stream arcs to your basin to represent water that accumulates along man-made objects such as roads. Roads often disrupt the natural flow of watersheds and water collects along roads just as it collects in a stream. This collected water needs to be "added" into your watershed in order to properly model the real-life situation.

2.3.1 Open Data Files

1. Select ***File* | *New***

2. Select *No* when asked to save changes

3. Select ***File* | *Open***

4. Locate the folder ***C:\Program Files\WMS81\tutorial\demedit***

5. Open *“trailmount.dem”*

6. Select *OK* on Importing USGS DEMs Dialog

7. Select ***Edit | Current Coordinates*** to set your current coordinates

8. Select *Global Projection*

9. Select *Set Projection*

10. Set the *Projection* to *UTM*, *Datum* to *NAD 27*, *Planar Units* to *METERS*, and *Zone* to *12 (114°W* **-** *108°W – Northern Hemisphere)*

11. Select *OK*

12. Set Vertical Units to *Meters*

13. Select *OK*

Skip section 2.3.2 if you are not able to connect to the Internet using your computer.

2.3.2 Getting a Background Image Using the TerraServer

Using an Internet connection we will now download the topographic map image directly from the TerraServer and open it in WMS.

1. Select the *Get Data* tool 

2. Drag a box around the extents of the rectangle as shown in Figure 2-6 to define the region of the topographic image

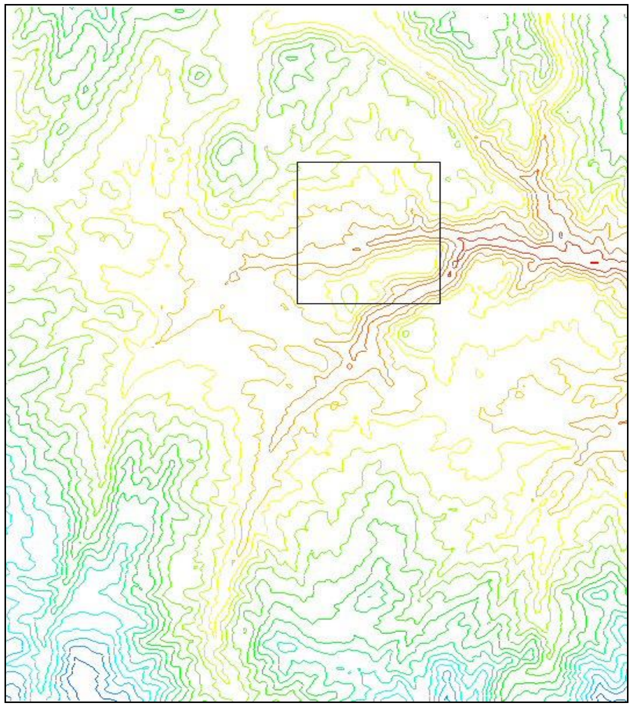


Figure 2-6: Define the region of the topographic image

3. Toggle on the *TerraServer topo* option
4. Select *OK* to start the downloading process
5. Enter *"TrailMtTopo"* and click *Save*
6. Click *Yes* to create the default wizard file name
7. Set the map scale to *2m*. It may take a few minutes to complete the downloading process.

WMS will automatically open the image after downloading it. If you were able to successfully complete all the steps in this section you can skip section 2.3.3.

2.3.3 Open TIF File

1. Select ***File* | *Open***
2. Open *"trailmountain.tif"*

2.3.4 Run TOPAZ

1. Switch to the *Drainage* module

2. Select ***DEM | Compute TOPAZ Flow Data***
3. Select *OK*
4. Select *OK*
5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
6. Right-click on *DEM* and select ***Display Options***
7. Change the Minimum Accumulation For Display to **0.06** mi^2
8. Select *OK*
9. Zoom in around the topographic image that you just downloaded from the TerraServer (or zoom in to the rectangle shown in Figure 2-6)

2.3.5 Basin Delineation

1. While in the Drainage module, select the *Create Outlet Point* tool
2. Click anywhere on the DEM to create an outlet
3. Select *OK* if you get a message telling you that the outlet is not located in a flow accumulation cell
4. Enter a Feature Point X-value of **375604.9** and a Feature Point Y-value of **4271336.7** in the Properties window on the right of the screen to edit the outlet location
5. Select ***DEM | Delineate Basins Wizard***
6. Select *OK* twice

2.3.6 Add Stream Arcs Along Road

1. Choose the *Zoom* tool
2. Zoom in along the road to the rectangle shown in Figure 2-7

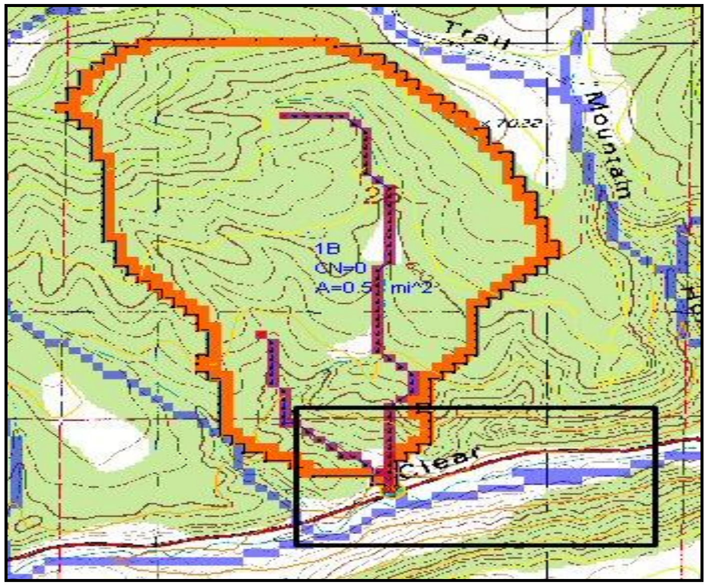


Figure 2-7: Zoom in to rectangle along road

3. Select the Drainage coverage in the Project Explorer

4. Choose the *Create Feature Arc* tool 

5. Select ***Feature Objects | Attributes***

6. Select the *Stream* option

7. Select *OK*

8. Create two stream arcs along the road as shown in Figure 2-8. Begin each arc by clicking on the node on the existing stream arc which is just upstream of the outlet, and end by double-clicking.

Only one basin is delineated by adding the stream arcs beginning at this node. It is possible to create the stream arcs beginning at the actual outlet point, but this causes multiple basins to be delineated, which may not be desirable. Also, to avoid creating basins on the south side of the highway, draw each arc on the north side of the flow accumulation lines. You are creating the stream arcs along the road to show that the water collected by the road drains into the same area the main watershed drains into.

Figure 2-8: Roadway collection stream arcs

2.3.7 Basin Delineation

1. Right-click on the *DEM* under the *Terrain Data* in the Project Explorer and select ***Display Options***
2. On the DEM tab, toggle on *Flow Direction* and *Points*; toggle off *Stream*, *Flow Accumulation*, *Color Fill Drainage Basins*, and *Fill Basin Boundary Only*
3. Select *OK*

Notice that flow directions arrows for DEM points are visible, but not necessarily for every DEM point because the display of flow directions is adaptive. You will see more flow directions as you zoom in and fewer flow directions as you zoom out.

4. Zoom in along the road until you can see flow direction arrows for every DEM point as shown in Figure 2-9

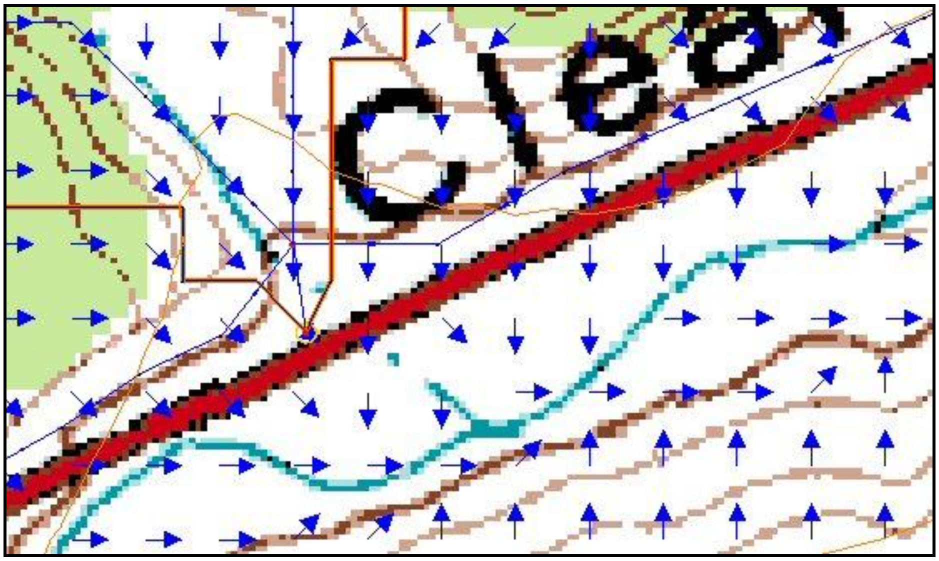


Figure 2-9: Flow direction arrows

The flow directions arrows show that flow moves right over the road and into the stream on the other side of the road. Watch how the flow directions change along that stream arcs that you added next to the road when you redefine the basins.

5. Switch to the *Drainage* module
6. Select ***DEM | Define Basins***
7. Select ***Display | Display Options***
8. On the DEM tab toggle off *Flow Direction* and *Points*; toggle on *Stream*, *Flow Accumulation*, *Color Fill Drainage Basins*, and *Fill Basin Boundary Only*
9. Select *OK*
10. Zoom out until you can see the extents of the drainage basin (HINT: Scroll up on the mouse wheel to zoom out)
11. Select ***DEM | Basins->Polygons***
12. Select ***DEM | Compute Basin Data***
13. Select *OK*

You can see that the drainage along the road has now been diverted into the original basin as shown in Figure 2-10.

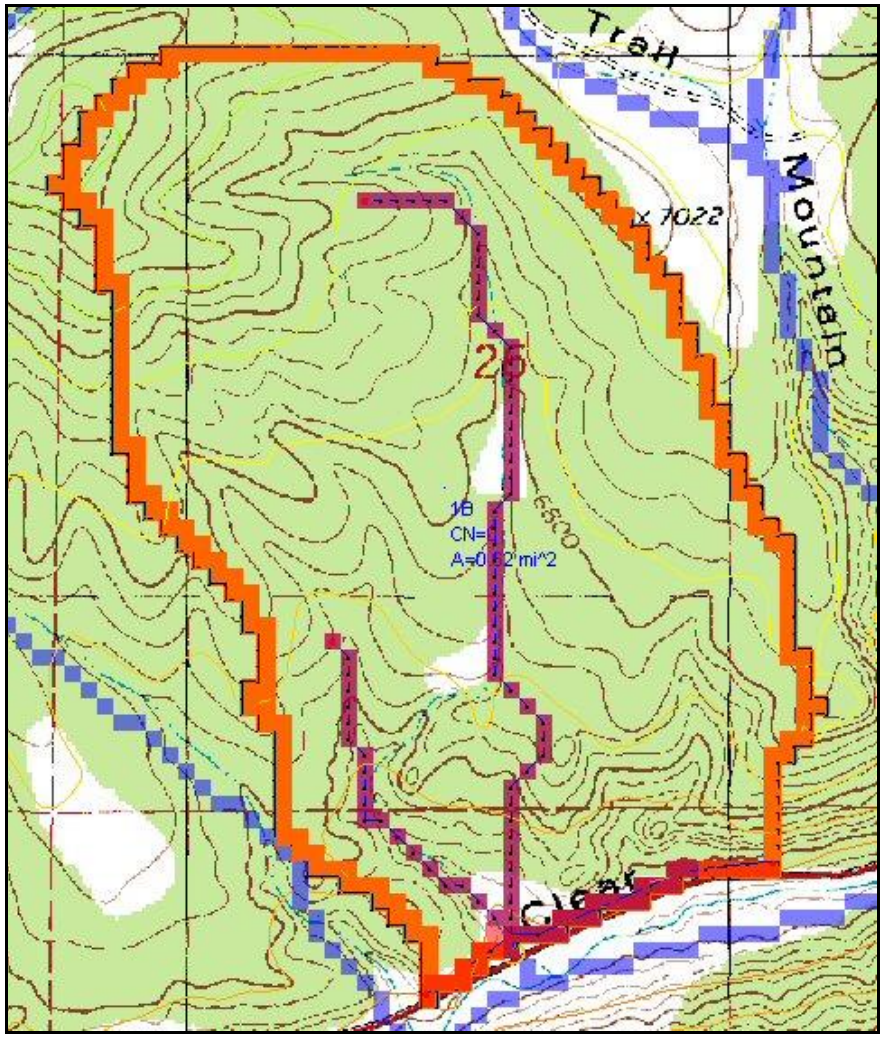


Figure 2-10: New basin including road stream arcs

2.4 Editing Flow Directions

Flow directions can be inaccurate due to imprecision in the DEM. The flow direction for each DEM point can be manually edited in order to improve accuracy in basin delineation or creation of stream arcs.

Notice that the flow accumulations show a stream that differs from the stream shown in the background image in Figure 2-11.

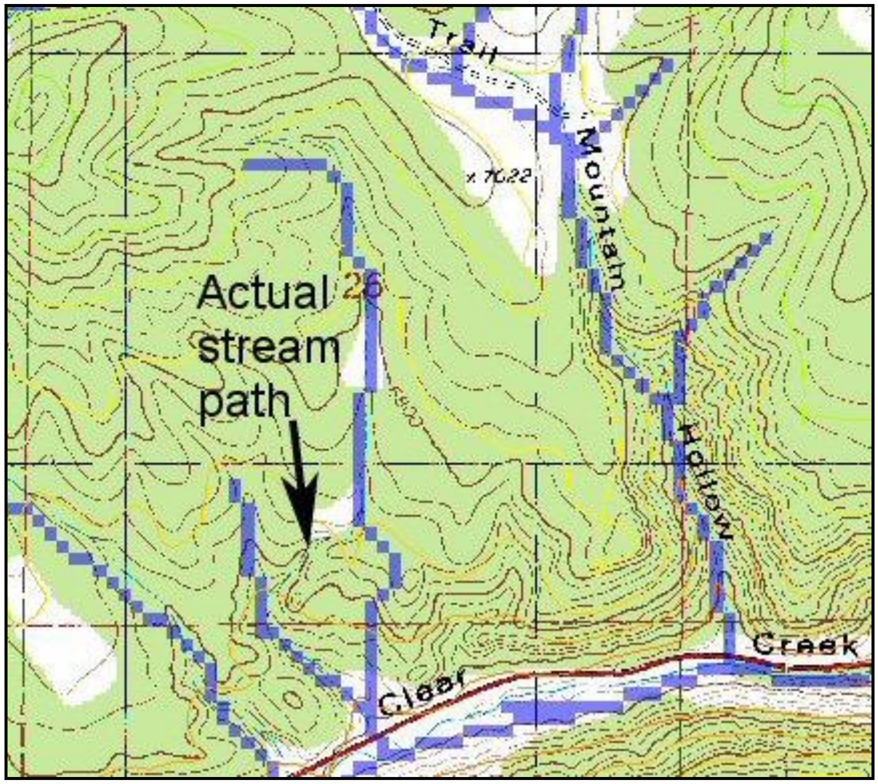


Figure 2-11: Stream path differences

2.4.1 Edit Flow Directions for DEM Points

Correct the flow directions for each of the numbered DEM cells in Figure 2-12.

1. Switch to the *Terrain Data* module
2. Zoom in to see the area shown in Figure 2-12

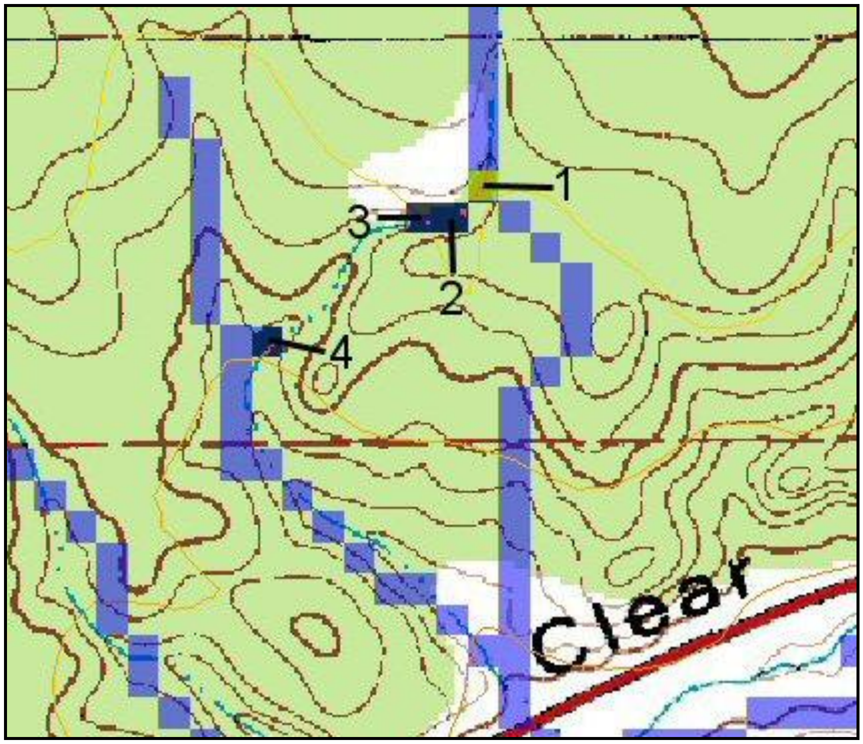


Figure 2-12: Edit flow directions for the numbered DEM cells

3. Use the *Select DEM Points* tool and double-click on a numbered cell shown in Figure 2-12. This will bring up the DEM Point Attributes dialog shown in Figure 2-13.

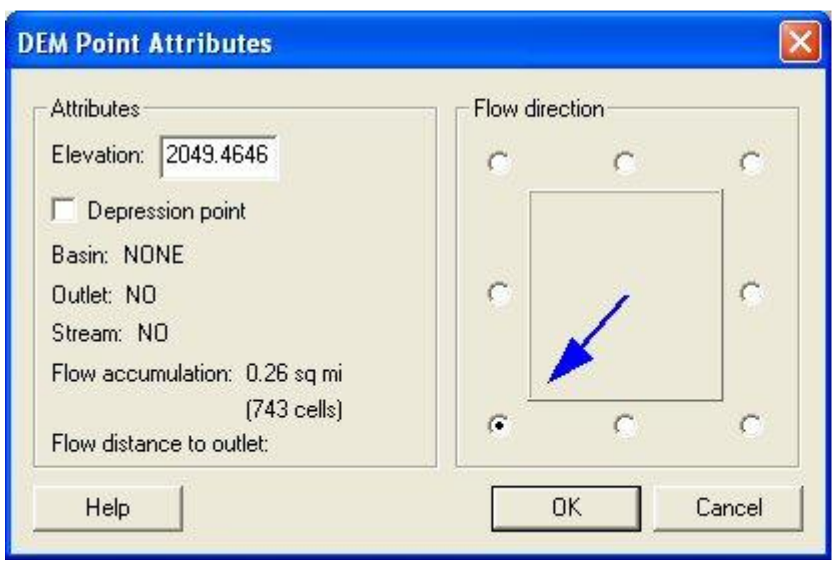


Figure 2-13: DEM Point Attributes dialog

4. Change the flow direction according to Table 2-1

Table 2-1: New flow directions

DEM Cell	New Flow Direction
1	↙
2	←
3	←
4	↙

5. Select *OK*

6. Toggle on *Compute flow accumulations* ONLY after the last flow direction has been edited

7. Select *OK*

8. Repeat Step 1-5 for all numbered DEM cells in Figure 2-12

It is possible to edit the flow direction for multiple DEM points simultaneously using the ***DEM | Point Attributes*** command in the *Terrain Data* module.

2.4.2 Basin Delineation

1. Switch to the *Drainage* module

2. Select ***DEM | Delineate Basins Wizard***

3. If prompted, select *OK* to delete all existing feature data and recreate using the new basin delineation

4. Select *OK*

5. Select *OK*

The stream path and the geometric calculations for the basin affected by the stream path are now correct.

2.5 Conclusion

In this exercise you have learned some of the advanced basin delineation features that set WMS apart from other GIS-based automated delineation techniques. While this exercise illustrated only a few ways these tools can be

applied, proper understanding enables you to use the tools for many different scenarios where the automated delineation does not yield the results you expect.

CHAPTER 3

Time of Concentration Calculations and Computing a Composite CN

This exercise will discuss tools that are helpful in calculating the time of concentration and in computing a composite curve number (CN). In particular, two models, the United States Geological Survey's (USGS) National Streamflow Statistics (NSS), and the National Resources Conservation Service's (NRCS) TR-55, will be discussed.

3.1 Opening the Drainage Basin

First we will open a WMS Project file (*.wpr) that contains a DEM that was previously downloaded from the Internet. A single watershed basin has been delineated from the DEM data and converted to feature objects.

1. Close all instances of WMS
2. Open WMS
3. Switch to the *Drainage* module
4. Select ***File | Open***
5. Locate the folder ***C:\Program Files\WMS81\tutorial\nss***
6. Open *"NSS_FL.wpr"*

3.2 Prepare the Basin for Use with NSS

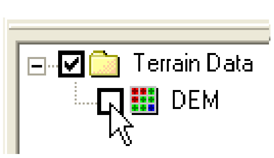
We will now use WMS to calculate the basin area, basin slope, and other parameters that can be used in conjunction with NSS.

1. Select ***DEM | Compute Basin Data***
2. Select the *Current Coordinates* button
3. Set the Vertical units to *Meters*
4. Select *Set Projection*
5. Select *METERS* in the *Planar Units* Field
6. Select *OK*
7. Select *OK*
8. Set the Basin Areas to *Square miles*
9. Set the Distances to *Feet*
10. Select *OK* to compute the parameters

In order to see the parameters that will be used with the NSS program, you can turn them on for display.

11. Select ***Display | Display Options*** 
12. Select *Drainage Data*
13. Check the display toggle for *Basin Slopes* (Basin areas should already be on)
14. Select *OK*

Basin attributes are displayed at the centroid of the basin. In order to see the parameters more clearly, turn off the display of the DEM



15. If needed, expand the Terrain Data folder in the Project Explorer
16. From the Project Explorer, toggle off the check box for the DEM

Your screen should now look like Figure 3-1

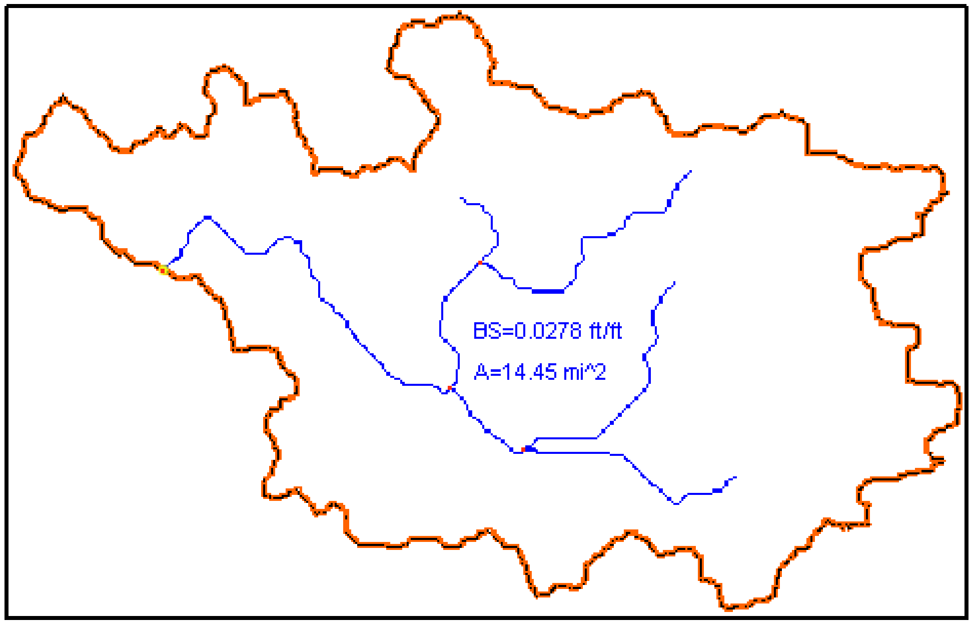


Figure 3-1: Drainage basin with parameters computed

3.3 Calculating Percentage of Lake Cover

The regression equation for Region B of Florida includes a parameter (LK) to define the ratio of the area of lakes in the basin to the total basin area (as a percent). We will use the Compute Coverage Overlay calculator in WMS to calculate the percentage of lake cover in our drainage basin. The only other parameter in the regression equation for Region B of Florida is drainage area (DA), something that is automatically computed using the Compute Basin Data command.

3.3.1 Opening the Land Use Coverage

In order to compute the percentage of lake cover in our watershed, we will read in land use data from a typical USGS land use file. Each polygon in the coverage is assigned a land use code that corresponds to a land use type. For this land use coverage, the codes for water bodies (lakes, reservoirs, wetlands) include 52, 53, 61, and 62. We will look for these codes to determine the value for LK.

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the coverage type to *Land Use*
4. Select *OK*

5. Right-click on *GIS layers* and select ***Add Shapefile Data***

6. Open *"valdosta.shp"* and make it the active layer

This land use shapefile was obtained from www.webgis.com, but the EPA and other websites contain similar information. Alternatively, we could have digitized land use polygons from an image (discussed in Volume 1, Chapter 3: Basic Feature Objects).

7. Choose the *Select Shapes* tool 

8. Drag a selection box around the drainage basin polygon

9. Select ***Mapping | Shapes -> Feature Objects***

10. Select *Next*

11. The LUCODE with the land use ID is automatically mapped so you can continue by selecting *Next*

12. Select *Finish*

13. Hide *valdosta.shp* by toggling off its check box in the Project Explorer (you may need to expand the GIS Layers folder to see it)

Only the portion of the shapefile that was selected will be used to create polygons in the Land Use coverage. The following figure displays the resulting land use polygons and their respective land use codes. This land use classification is consistent among all of the USGS land use data, were codes from 10-19 are urban, 20-29 agricultural, etc. A complete listing of code values can be found in the WMS Help file.

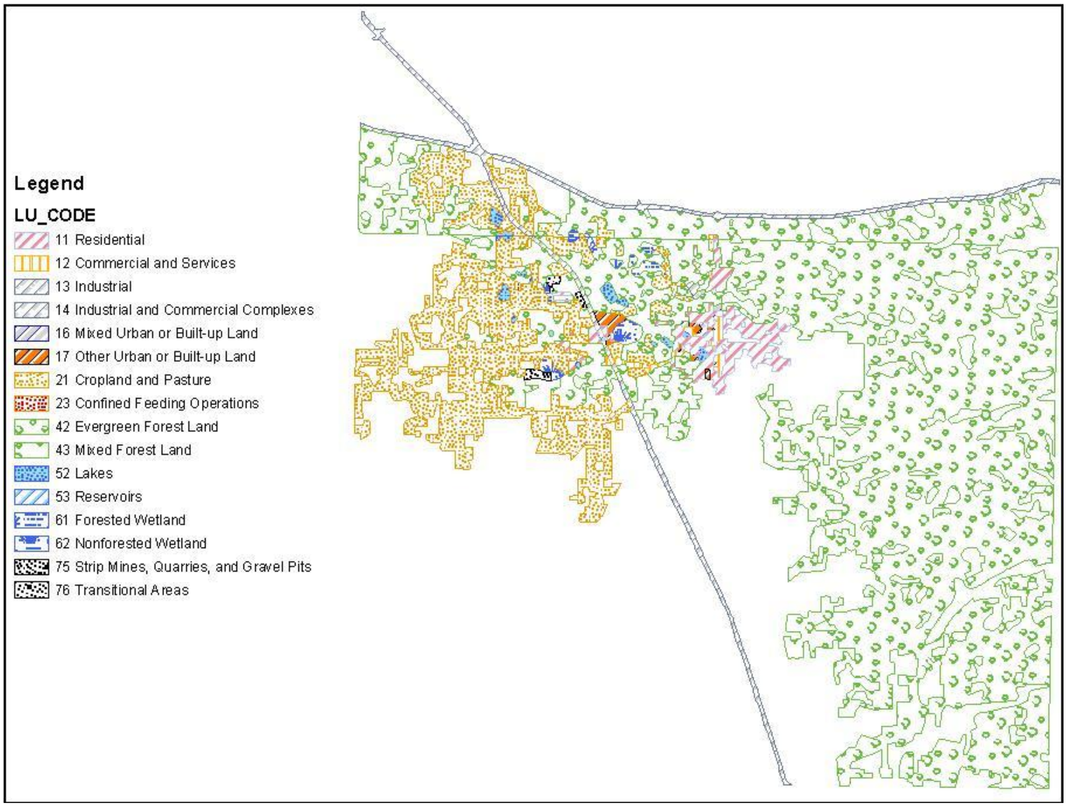
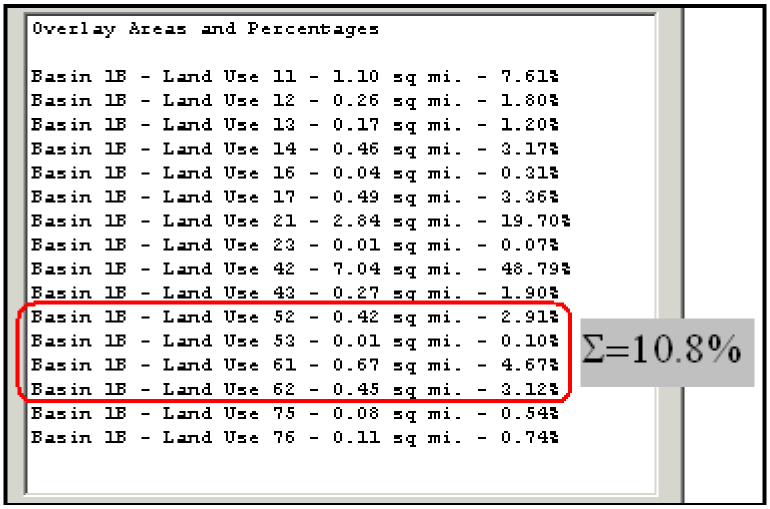


Figure 3-2: Land use codes used in Valdosta.shp.

3.3.2 Using the Compute Coverage Overlay Calculator

1. Switch to the *Hydrologic Modeling* module 
2. Select ***Calculators | Compute Coverage Overlay***
3. Make sure that *Drainage* is chosen as the Input Coverage
4. Make sure that *Land Use* is set as the Overlay Coverage
5. Select the *Compute* button

According to the USGS land use classification, code values in the 50's and 60's represent water bodies. To obtain the value for LK, we sum together the computed overlay percentages for Land Uses 52, 53, 61, and 62, as shown in Figure 3-3.



```
Overlay Areas and Percentages

Basin 1B - Land Use 11 - 1.10 sq mi. - 7.61%
Basin 1B - Land Use 12 - 0.26 sq mi. - 1.80%
Basin 1B - Land Use 13 - 0.17 sq mi. - 1.20%
Basin 1B - Land Use 14 - 0.46 sq mi. - 3.17%
Basin 1B - Land Use 16 - 0.04 sq mi. - 0.31%
Basin 1B - Land Use 17 - 0.49 sq mi. - 3.36%
Basin 1B - Land Use 21 - 2.84 sq mi. - 19.70%
Basin 1B - Land Use 23 - 0.01 sq mi. - 0.07%
Basin 1B - Land Use 42 - 7.04 sq mi. - 48.79%
Basin 1B - Land Use 43 - 0.27 sq mi. - 1.90%
Basin 1B - Land Use 52 - 0.42 sq mi. - 2.91%
Basin 1B - Land Use 53 - 0.01 sq mi. - 0.10%
Basin 1B - Land Use 61 - 0.67 sq mi. - 4.67%
Basin 1B - Land Use 62 - 0.45 sq mi. - 3.12%
Basin 1B - Land Use 75 - 0.08 sq mi. - 0.54%
Basin 1B - Land Use 76 - 0.11 sq mi. - 0.74%
```

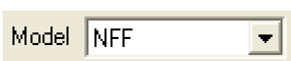

Figure 3-3: Summing the percentages of the codes representing water cover

The Coverage Overlay command can be used in a similar fashion to determine the percentage of forested areas (codes in the 40's), or any other classification type in a land use file, or a soil file.

6. Select *Done*

3.4 Running NSS

The geometric data computed from the DEM has automatically been stored with the NSS data. You can now run a simulation using the derived data.



1. Make sure that the Model combo box is set to *NSS*
2. Select the *Frame* macro 
3. Select the *Select Basin* tool 
4. Double-click on the basin icon for Basin 1B
5. Choose *Florida* from the list of states
6. Highlight *Region B* from the list of Regional regression equations
7. Select the *Select->* button to move Region B to the Selected Equations window
8. Enter **10.8** for the Lake Area variable (you may have to scroll the Variable Values window in order to see the Lake Area variable)
9. Select the *Compute Results* button

The peak flow (Q) values are displayed in the window at the bottom of the dialog.

3.4.1 Exporting the Flow Data

Once flow data is computed it may be exported to a text file in the format shown in the window, along with pertinent information used in computing the peak flow values.

1. Select the *Export* button
2. Locate a directory, and define a name for the file
3. Select *Save*

The exported file can be viewed using any word processor, or inserted into a separate report document.

3.5 Time Computation / Lag Time Calculation

The NSS program provides a way to determine an "average" hydrograph based on the computed peak flow and a basin lag time. A dimensionless hydrograph is used to define a basin hydrograph for the watershed based on the computed peak flow.

1. Scroll down in the Results window if necessary, and select the line of text corresponding to a Recurrence [years] of 50
2. Select the *Compute Hydrograph* button
3. Select the *Compute Lag Time - Basin Data* button
4. Change the Method combo box to the *Custom Method* (the very last one in the list)
5. Select *OK*

The computed lag time in minutes is shown in the lag time edit field. Time of concentration equations can also be used to calculate the basin lag time. WMS will convert the time of concentration to lag time by the equation: $T_{lag} = 0.6*T_c$

6. Select the *Compute Lag Time – Basin Data* button
7. Change the Computation type combo box to *Compute Time of Concentration*
8. Change the Method combo box to the *Kerby Method for overland flow*

9. Select *OK*

Note the difference in the calculated lag time between the two methods. These two equations, along with the other available options in the Basin Time Computation calculator, can be used to estimate the lag time of the basin. Compare the results of the different equations available to best describe the characteristics of the basin.

10. Select *OK*

11. Select the *Done* button

A hydrograph icon will appear next to the basin icon for Basin 1B. You can examine the hydrograph in more detail:

12. Select the *Select Hydrograph* tool 

13. Double-click on the hydrograph icon

The hydrograph is displayed in the Graphics window.

14. Close the hydrograph plot window by clicking on the X in the upper right corner of the window

15. Select ***File | New***

16. Select *No*

3.6 Using TR-55 to Compute Tc and CN

Travel times (time of concentration, lag time, and travel time along a routing reach) are critical to performing analyses with any of the hydrologic models. You will learn of two different ways WMS can be used to compute time of concentration for a TR-55 simulation (lag times are computed in the same way):

- Runoff distances and slopes for each basin are automatically computed whenever you create watershed models from TINs or DEMs and compute basin data. These values can then be used in one of several available equations in WMS to compute lag time or time of concentration

- If you want to have a little more control (and documentation) over the lag time or time of concentration, you will use a time computation coverage to define critical flow paths. Time computation coverages contain flow path arc(s) for each sub-basin. An equation to estimate travel time is assigned to each arc and the

time of concentration (or lag time) is the sum of the travel times of all arcs within a basin. Lengths are taken from the length of the arc and slopes derived if a TIN or DEM are present.

In this exercise you will compute the time of concentration for the two sub-basins and the travel time between outlet points in the watershed shown below. You will use the TR-55 library of equations, but you could just as easily use one of the other pre-defined equations, or enter your own equation.

3.7 Reading a TR-55 Project

You will first read in a project file of an urban area that has been processed and delineated as a single basin. The project includes a drainage coverage, a time computation coverage, and two shapefiles for the land use and soil type data.

1. Switch to the *Map* module
2. Select ***File | Open***
3. Locate the folder ***C:\Program Files\WMS81\tutorial\tr-55***
4. Open *"suburbtr55.wpr"*

3.8 Assigning Equations to Time Computation Arcs

A flow path arc has already been defined for the basin. This arc represents the longest flow path for the urban area, starting from a sandy area at the top of the basin, following along the streets and down towards a detention pond at the bottom of the basin. The arc has been split into four different segments to assign different equations to determine the travel time for the arc. Use the following figure as a guide while defining the equations.

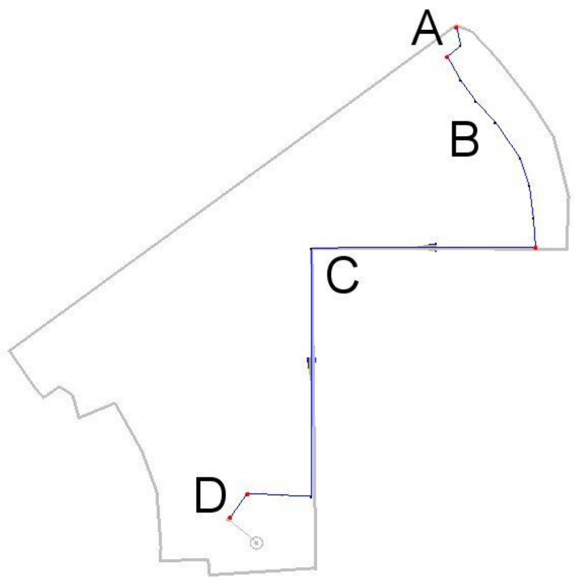


Figure 3-4: Time Computation Arcs.

1. Switch to the *Time Computation* coverage

2. Choose the *Select Feature Arc* tool 

3. Double-click on the arc labeled A in Figure 3-4

By default the arc will be a TR-55 sheet flow equation arc, so all you need to do is define the overland Manning's roughness coefficient and the 2yr-24hr rainfall. Length and slope will already be entered (from the selected arc).

4. Click on the Manning's line in the Variables text window

5. Enter a value of **0.03** in the Variable value edit window

6. Click on the 2 yr - 24 hr rainfall line in the Variables text window

7. Enter a value of **1.1**

You should notice in the Instructions/Results window that you are told what variables need to be defined before a travel time can be computed. Once you have entered all the necessary values, this same window reports the travel time for this arc. In this way, you can compute travel time for any arc segment no matter what the application is.

8. Select *OK*

You have now defined an equation for the overland sheet flow segment in the basin and you are ready to define the next segments as shallow concentrated flow.

9. Double-click on the arc labeled B in Figure 3-4
10. Change the equation type to *TR-55 shallow conc eqn*
11. Click on the Paved line in the Variables text window
12. Enter *"yes"* in the Variable value edit window
13. Select *OK*
14. Repeat for the arc labeled D, using the same equation type. In this case set the Paved value to *"no"*

The remaining arc will be defined as an open channel flow arc.

15. Double-click on the arc labeled C in Figure 3-4
16. Change the arc type to *TR-55 Open channel eqn*
17. Click on the Manning's n line
18. Enter a value of **0.017** in the Variable value edit window
19. Select the hydraulic radius line in the Variables window
20. Select the *Hydraulic Radius* button to open up the channel calculator so that the hydraulic radius can be computed from estimates of the curb in the subdivision
21. Change the Channel type to *Triangular*
22. Enter a longitudinal slope of **0.010** ft/ft
23. Enter a Side slope 1 (Z1) of **10**
24. Enter a Side slope 2 (Z2) of **0.01**
25. Choose the *Enter depth* option
26. Enter a depth of **0.5** (an approximated depth since we do not know what the flow is at this point)
27. Select the *Calculate* button
28. Select *OK* for both dialogs

You have now defined the necessary parameters for computing travel time using the TR-55 open channel flow (Manning's) equation. If you wish you can continue to experiment with the channel calculator to compute the hydraulic radius rather than entering the given values.

You now have defined equations and variable values for each flow path segment. You can change these equations and variables, add new flow path segments, etc. in order to determine the best flow paths and most appropriate equations for each basin. In other words, the process is subjective and it may take a few iterations to get the best value.

3.9 Computing Time of Concentration for a TR-55 Simulation

Before assigning time of concentrations to each basin you need to decide which model you want to use. For this exercise you will be running TR-55, but the same time computation tools you learn in this exercise could be used for any of the supported WMS models (such as in the TR-20 basin data dialog, the HEC-1 Unit Hydrograph method dialog, and the Rational Method dialog).

1. Select the *Hydrologic Modeling* module
2. Change the Model drop-down list to *TR-55*
3. Select the *Select Basin* tool
4. Select the basin
5. Select ***TR-55* | *Run Simulation***

In the TR-55 dialog, notice the two drop-down boxes at the top. These provide the ability of changing TR-55 information for basins and outlets individually or collectively.

6. Enter a Rainfall value of **1.5**
7. Change the Rainfall distribution to *Type II*
8. Select the *Compute Tc - Map Data* button. You will see the four time computation arcs that are in the basin
9. You can create a detailed report as a text file if you want by selecting the *Export Data* or *Copy to Clipboard* buttons
10. Select *Done*
11. Select *OK*

The sum of the travel times for these arcs will be used as the time of concentration for this basin

Note that you could bring up the time computation attributes dialog and change the equation or any of the equation variables by selecting the *Edit Arcs* button.

3.10 Computing a Composite Curve Number

In this part of the exercise, you will learn how to overlay land use and soil coverages on your delineated watershed in order to derive a curve number (CN).

3.10.1 Land Use Table

Now you need to create a land use table with IDs and CNs for each type of land use on your map. A table has been provided, but it is incomplete. To finish the table with all of the IDs and CNs for the shapefiles in the project, or just to edit the table in general, complete the following steps:

1. Select ***File | Edit File***
2. Open *landuse.tbl*
3. If prompted, choose a text editor to edit the file with, by choosing Notepad or another favorite editor in the Open With drop down list, and select *OK*

In the text editor, you will find three lines of text listing three IDs along with their CN values. The file format for this file is an ID value, followed by a comma, the name of the land use ID in quotation marks, followed by a comma, followed by the comma separated CN values for soil types A, B, C, and D, respectively. This file includes CN values for landuse types “Transportation, Communications”, “Other Urban or Built-Up Land”, and “Bare Ground”. The landuse shapefile in this project also contains landuse polygons for residential areas, with an ID for 11. Complete the land use table by editing the file:

4. Add the following line to the file: *11, "Residential", 61, 75, 83, 87*
5. Save the file and close the editor

3.10.2 Computing Composite Curve Numbers

In order to compute composite curve numbers, WMS needs to know which type of soil underlies each area of land. You will need either a landuse and soil

type coverage, or a landuse and soil type shapefile with the appropriate fields. For this exercise, we will be using landuse and soil type shapefiles.

1. Select ***Calculators | Compute GIS Attributes***

2. Make sure the *SCS Curve Numbers* option is selected in the Computation section of the dialog

3. Select *GIS Layers* in the Using section of the dialog

4. Select the Soil Layer Name to be *soils_poly.shp*

5. Make sure the Soil Group Field has been set to *HYDGRP*

6. Select the Land Use Layer Name to be *landuse_poly.shp*

7. Make sure the Land Use ID Field has been set to *LU_CODE*

You may have your land use and soil type tables stored in data files, such as the one you previously edited. Instead of manually assigning the data as you have here, you would read these tables in from this dialog using the Import button.

Whether you have manually created tables or read them in from files, you should see the land use IDs and CNs for each soil type, and land use descriptions in the window of the Mapping section.

8. Select the *Import* button near the bottom of the dialog

9. Find and open the mapping table *"landuse.tbl"*

You should now see the assignment of CN values for the land use table previously edited.

10. Select *OK* to compute the composite CNs

A Runoff Curve Number Report is generated and opened automatically. The composite curve number appears at the bottom of the report.

11. Select the *Select Basin* tool 

12. Double-click on the basin

Notice that the Curve number edit field has been updated with the calculated value from the Compute GIS Attributes dialog.

3.11 More TR-55

While you were entering the data for the basin you may have noticed that instructions are given in the TR-55 data window to let you know what must be entered before a peak Q can be determined. Once you enter all of the data the peak Q is computed and displayed in the same window. You can also get help for anything listed in the window.

1. Notice that the TR-55 reference equation for computing peak flow is displayed next to Peak Discharge

2. Select the *Compute Hydrograph(s)* button

3. Select *OK* to close the TR-55 dialog

4. Choose the *Select Hydrograph* tool 

5. Double-click on the hydrograph icon that is displayed by the upper basin to view the hydrograph in a separate window

3.12 Conclusion

This completes the chapter on using the time computation coverage to compute time of concentration and travel times and the land use and soil coverage to compute a composite CN value. In the process you have also learned about the TR-55 interface.

CHAPTER 4

HEC-1 Interface

WMS has a graphical interface to HEC-1. Geometric attributes such as areas, lengths, and slopes are computed automatically from the digital watershed. Parameters such as loss rates, base flow, unit hydrograph method, and routing data are entered through a series of interactive dialog boxes. Once the parameters needed to define an HEC-1 model have been entered, an input file with the proper format for HEC-1 can be written automatically. Since only parts of the HEC-1 input file are defined in this chapter, you are encouraged to explore the different available options of each dialog, being sure to select the given method and values before exiting the dialog.

The US Army Corps of Engineers now supports HMS rather than HEC-1, but the hydrologic calculations for the options within HEC-1 have not changed. Results of the two models will be identical.

4.1 Objectives

As a review, you will delineate a watershed from a DEM. You will then develop a simple, single basin model using the delineated watershed to derive many of the parameters. Land use and soil shapefiles (downloaded from the Internet) will be used to develop a SCS curve number (CN) value. After establishing the initial HEC-1 model, other variations will be developed, including defining multiple basins with reach routing and including a reservoir with storage routing.

4.2 Delineating the Watershed

Since the land use, soil type, and DEM data for our watershed all originate in the Geographic coordinate system, we will begin by opening them together and converting them to UTM coordinates. The land use and soil type data were downloaded from the Environmental Protection Agency (EPA) website. The DEM data used for this watershed were previously downloaded from the National Elevation Dataset website as was demonstrated in the DEM Basics exercise (Volume 1, chapter 4).

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open*** 
4. Locate the folder ***C:\Program Files\WMS81\tutorial\hec-1***
5. Select *NED GRIDFLOAT Header (*.hdr)* from the Files of type list of file filters
6. Open *"67845267.hdr"*
7. Select *OK*
8. When prompted if you want to convert the current coordinates select *No*

4.2.1 Create Land Use and Soil Coverages

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the coverage type to *Land Use*
4. Select *OK*
5. Create a new coverage once again and set its coverage type to *Soil Type*

4.2.2 Open the Soils Data

1. Make sure the *Soil Type* coverage is active in the Project Explorer
2. Right-click on *GIS layers* in the Project Explorer and select ***Add Shapefile Data***

3. Open *"statsgo.shp"*

4. Right-click on *statsgo.shp* layer in the Project Explorer

5. Select ***Open Attribute Table***

Notice that the table has three fields named AREA, PERIMETER, and MUID

6. Select *OK*

4.2.3 Join Soils Database File Table to Shapefile Table

1. Right-click on *statsgo.shp* in the Project Explorer

2. Select ***Join Table to Layer***

3. Open *"statsgoc.dbf"*

4. Ensure that Shapefile Join Field and Table Join Field are both set to *MUID*

5. Change the Table Data Field to *HYDGRP*

6. Select *OK*

7. Right-click on *statsgo.shp* in the Project Explorer

8. Select ***Open Attribute Table***

Notice that the HYDGRP field is now a part of the shapefile.

9. Select *OK*

4.2.4 Convert Soil Shapefile Data to Feature Objects

1. Choose the *Select Shapes* tool 

2. Draw a selection box around the DEM extents

3. Select ***Mapping | Shapes -> Feature Objects***

4. Select *Next*

This window shows all of the attribute fields in the soils shape file. Because this file was derived from a standard NRCS statsgo file you will notice that the hydrologic soil groups field is named HYDGRP and so WMS will automatically map this to be the soil type. If the attribute field were named

anything other than HYDGRP then you would have to manually map it using the drop down list in the spreadsheet.

5. Make sure the HYDGRP field is mapped to the SCS soil type attribute
6. Select *Next*
7. Select *Finish*
8. Clear the selected polygons by single-clicking somewhere beyond the extents of the shapefile polygons
9. Hide the *statsgo.shp* file by toggling off its check box in the Project Explorer

4.2.5 Open the Land Use Data

1. Select the *Land Use* coverage in the Project Explorer to designate it as the active coverage
2. Right-click on *GIS layers* in the Project Explorer and select ***Add Shapefile Data***
3. Open *"l_richut.shp"*
4. Choose the *Select Shapes* tool 
5. Draw a selection box around the DEM extents
6. Select ***Mapping | Shapes -> Feature Objects***
7. Select *Next*
8. Make sure the LUCODE field is mapped to the Land use attribute
9. Select *Next*
10. Select *Finish*
11. Hide the *l_richut.shp* file by toggling off its check box in the Project Explorer

4.2.6 Convert/Set the Coordinate System of the Data

1. Right-click on *Terrain Data* in the Project Explorer and select ***Coordinate Conversion***
2. Under the *Current Projection* Section toggle on *Specify*

3. Make sure *Global Projection* is toggled on and click *Set Projection*
4. Set *Projection* to *Geographic*, and *Datum* to *NAD 83*
5. Select *OK*
6. Set the *Datum* to *NAVD 88(US)* and the vertical units to *Meters*
7. In the *New Projection* section select *Global Projection*
8. Click *Set Projection*
9. Set *Projection* to *UTM*, *Datum* to *NAD83*, *Planar Units* to *METERS*, and *Zone* to *12 (114°W - 108°W – Northern Hemisphere)*
10. Select *OK*
11. Set the *Datum* to *NAVD 88(US)* and the vertical units to *Meters*
12. Click *OK*
13. Since we will not be using them until later, hide the *Land Use* and *Soil Type* coverages by toggling off their check boxes in the Project Explorer
14. Select the *Drainage* coverage from the Project Explorer to make sure it is the active coverage

4.2.7 Delineate the Watershed

1. Select the *Drainage* module 
2. Select the *Frame* macro 
3. Select ***DEM | Compute TOPAZ Flow Data***
4. Select *OK*
5. Select *OK* in the Units dialog
6. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
7. Select the *Zoom* tool 
8. Zoom in by dragging a box as illustrated in Figure 4-1

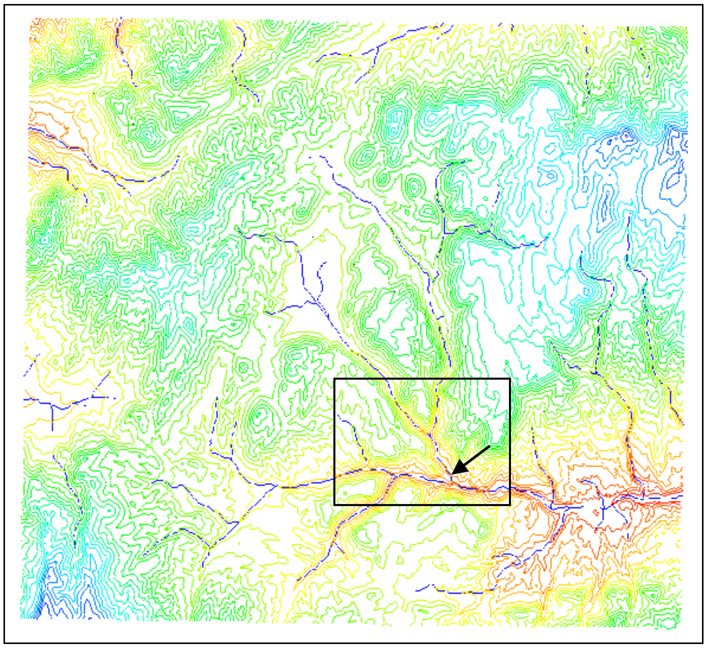


Figure 4-1: Zoom in on the area bounded by the rectangle above

9. Select the *Create Outlet Point* tool

10. Create a new outlet point where the tributary you just zoomed in on separates from the main stream as illustrated by the arrow in Figure 4-1. Make certain that the outlet point is on the tributary and not part of the main stream. Also, the outlet needs to be inside one of the flow accumulation (blue) cells. WMS will move the outlet to the nearest flow accumulation cell if you do not click right in one of the flow accumulations cells.

11. Select the *Frame* macro

12. Select ***DEM | Delineate Basins Wizard***

13. Select *OK*

14. Select *OK*

You have now completed the delineation of a single watershed. In order to make the view clearer for defining the hydrologic model you can turn off many of the DEM and other display options.

15. Right-click on DEM in the Project Explorer and select ***Display Options***

16. On the DEM tab, toggle off the display for *Watershed*, *Stream*, *Flow Accumulation*, and *DEM Contours*

17. On the Map tab, toggle *Vertices* off

18. Select *OK*

4.3 Single Basin Analysis

The first simulation will be defined for a single basin. You will need to enter the global, or Job Control parameters as well as the rainfall event, loss method, and unit hydrograph method.

4.3.1 Setting up the Job Control

Most of the parameters required for a HEC-1 model are defined for basins, outlets, and reaches. However, there are many "global" parameters that control the overall simulation and are not specific to any basin or reach in the model. These parameters are defined in the WMS interface using the Job Control dialog.

1. Switch to the *Hydrologic Modeling* module 

2. HEC-1 should be the default model, but if it is not select it from the drop down list of models found in the Edit Window

3. Select ***HEC-1 | Job Control***

4. The first three lines are for comments/identification at the top of the HEC-1 input file. The first line already has information indicating that the input file is generated by WMS (you can change this if you want). Enter *"Clear Creek Tributary Watershed"* for the second ID line. Enter your name and current date in the third line.

5. Leave the Day, Month, and Year fields alone

HEC-1 allows you to enter a date, but almost always you are simulating some kind of hypothetical or design storm and not an actual storm. If you change the simulation date you will need to be careful to make sure the storm date is in synch, but if you leave it alone, there will not be a problem.

6. Enter **5** (minutes) for the Computation time interval, and **300** for the Number of hydrograph ordinates. Leave the Beginning time at **0**.

An HEC-1 simulation will run for a length of time equal to the time step multiplied by the number of ordinates. If you are simulating a 24-hr storm but only run the simulation for 12 hours you will not capture the full hydrograph. Conversely, if you run a 24-hr simulation for 96-hrs you are probably going to have a lot of runoff ordinates equal to 0 at the end. In this case we are running

the simulation for 1500 minutes (slightly more than 24 hours) with an ordinate on the hydrograph being computed for every 5 minutes.

7. Set the computation units to *English units* (this should be the default)

Setting the computation units DOES NOT cause any units conversion to take place. You are simply telling HEC-1 that you will provide input units in English units (sq. miles for area, inches for rain, feet/miles for length) and expect results of computation to be in English units (cfs). If you specify Metric then you must ensure that input units are metric (sq. kilometers, mm for rain, meters/kilometers for length) and results will be in metric (cms).

8. Select *OK*

For now we will leave the other Job Control settings at their default values.

4.3.2 Setting up the Basin Data Parameters

In the first simulation you will treat the entire watershed as a single basin.

1. Select the *Select Basin* tool 
2. Double-click on the brown basin icon labeled 1B. Double-clicking on a basin or outlet icon always brings up the parameter editor dialog for the current model (in this case HEC-1)
3. Select the *Basin Data* button
4. Notice that the area has been calculated (in this case in sq. miles because we are performing calculations in English units)
5. Change the name to *CCTrib*. HEC-1 will only use the first SIX characters so do not use names longer than six characters for basins or outlets.
6. Select *OK*
7. Select the *Precipitation* button
8. Select the *Basin Average* option
9. Enter **1.8** (inches) for the Average precipitation depth
10. Select the *Define Series* button

In order to simulate a rainfall event you must enter both a rainfall depth and a temporal distribution. The SCS uses standard time distributions for different areas of the U.S. These are stored in WMS. You could also define your own

series according to an actual storm, or a design storm from a regulating agency.

11. In the Selected Curve drop down list select the *typeII-24hour* curve

12. Select *OK*

13. Select *OK*

14. Select the *Loss Method* button

15. Enter a Curve Number (CRVNBR field) of **70**. We will compute a CN value from actual land use and soil files later.

16. Select *OK*

17. Select the *Unit Hydrograph Method* button

18. Make sure the *SCS dimensionless* option is chosen (it is the default)

19. Select the *Compute Parameters - Basin Data* button

20. Set the Computation Type to *Compute Lag Time* (the default)

21. Set the Method drop down list to *SCS Method* (near the bottom of the list)

22. Select *OK* to update the computed lag time for the SCS dimensionless method

23. Select *OK*

24. Select *Done*

You now have all of the parameters set to run a single basin analysis.

4.3.3 Running HEC-1

Whenever you run a HEC-1 simulation, WMS will first save a standard HEC-1 input file. You will also be prompted for the name of an output file and a solutions file. The output file is the standard text output file generated by HEC-1 and the solution file is a plot file that contains the hydrographs formatted in a way that makes it easy for WMS to read and plot (it is actually the HEC-1 TAPE22 file).

1. Select ***HEC-1 | Run Simulation***

2. Click the browse button  next to the Input File

3. For the file name enter “*CCTrib*” and click *Save* (this specifies the file name but does not actually save it)

4. Verify that the *Save file before run* is toggled on

5. Select *OK*

6. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

The solutions will automatically be read in and you should see a small hydrograph plot to the upper right of the Basin icon (now labeled as CCTrib).

7. Double-click on the hydrograph icon.

A plot window will appear with the hydrograph. You will see that the hydrograph suddenly stops at 1500 minutes (the duration of the simulation as established in the Job Control dialog), but the simulation obviously has not run to completion.

8. Close the plot window by selecting the X in the upper right corner of the window

9. Select ***Hydrographs | Delete All***

10. Select ***HEC-1 | Job Control***

11. Set the Number of hydrograph ordinates to be **400**

12. Select *OK*

13. Select ***HEC-1 | Run Simulation***

14. Select *OK* (you can let it overwrite the other files)

15. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

16. Double-click on the hydrograph icon

You now have a completed HEC-1 simulation for a single basin and the resulting hydrograph should look something like the solution shown in Figure 4-2.

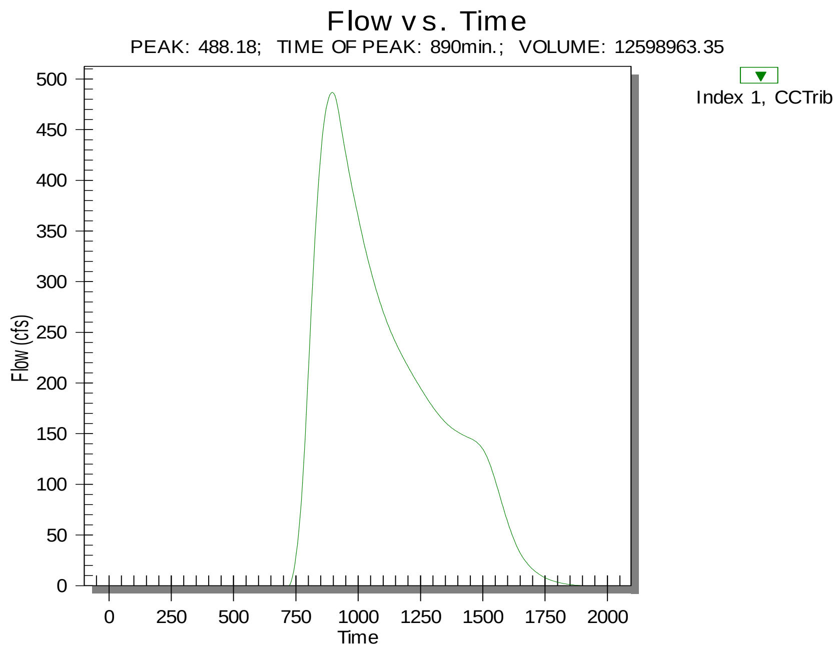


Figure 4-2: Solution hydrograph for HEC-1 simulation

17. Close the hydrograph window by selecting the X in the upper right of the window

4.4 Computing the CN Using Land Use and Soils Data

In the initial simulation you estimated a CN, but with access to the Internet it is simple to compute a composite CN based on digital land use and soils files. This was demonstrated in more detail in the Advanced Feature Objects exercise (Volume 1, chapter 6), but you will go through the steps here as a review.

4.4.1 Computing a Composite CN

In addition to the digital land use and soils file that overlap the watershed, you must have a table defined that identifies CN values for each of the four different hydrologic soil groups (A, B, C, D). This is described in detail at the gsda website (http://www.xmswiki.com/wiki/GSDA:GSDA), and in the Advanced Feature Objects exercise (Volume 1, chapter 6). For this exercise you will read in an existing file (you can examine it in a text editor if you wish) and compute the CN numbers.

1. Select the *Hydrologic Modeling* module 

2. Select ***Calculators | Compute GIS Attributes***

3. Select the *Import* button to load the mapping table

4. Select *OK* to overwrite the current definition

5. Find and open the file named *“scsland.tbl”*

6. Select *OK* to compute the CN from the land use and soils layers

You should find that CN computed from the land use and soils digital data is about 72 or 73. While there is still some “judgment” required in setting up the mapping table, there is a lot more justification for this value than the one previously estimated.

7. Close the Curve Number Report

4.4.2 Running HEC-1

You can now run another simulation to compare the results with the modified CN value.

1. Select ***HEC-1 | Run Simulation***

2. Select *OK* (it is fine to overwrite the existing files, but you can change the file names if you want)

3. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

4. Double-click on the hydrograph icon to plot both the old and the new hydrograph in a plot window

With the increased CN value you should see that the resulting hydrograph peaks higher (more runoff). The peak should be about 600 cfs rather than the 500 cfs that was generated with a CN value of 70.

5. Close the hydrograph window by selecting the X in the upper right corner of the window.

6. Select ***Hydrographs | Delete All***

4.5 Adding Sub-basins and Routing

You will now subdivide the watershed into two upper basins and one lower basin and define routing for the reaches that connect the upper basins to the watershed outlet.

4.5.1 Delineating the Sub-basin

1. Select the *Drainage* module
2. Select the *Zoom* tool
3. Create a zoom box around the region identified by a box in Figure 4-3

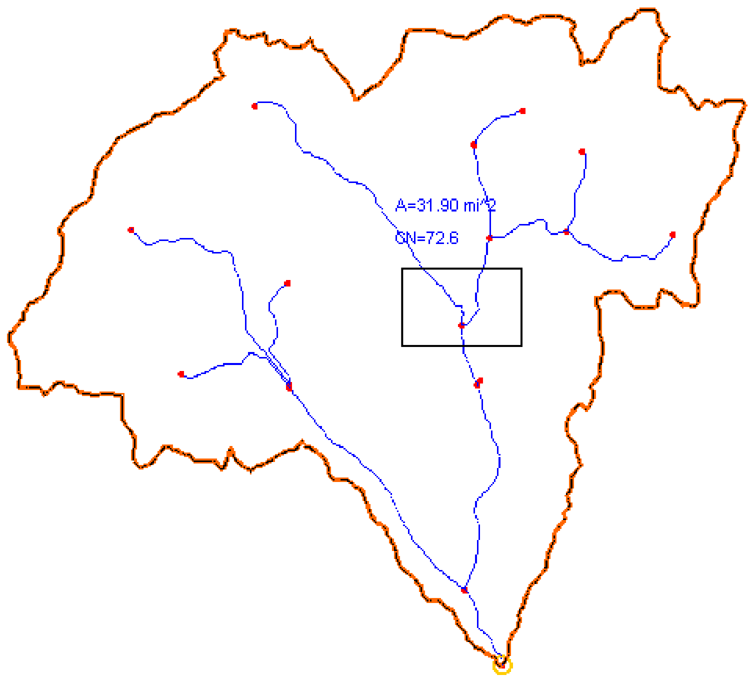
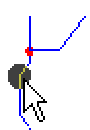


Figure 4-3: Zoom in on the area indicated by the rectangle.

4. Select ***Display* | *Display Options***
5. On the Map tab toggle on *Vertices*
6. Select *OK*
7. Select the *Select Feature Vertex* tool
8. Select the vertex that is just below the main branching point you just zoomed in around



9. Select ***DEM* | *Node* <-> *Outlet***

You created the outlet point just below the branch in order to have a single upstream basin. If you wanted a separate basin for each upstream branch you could define the branching node to be an outlet. Thus, WMS would automatically assume that you want separate basins for each branch, so we

have created a node just downstream of the branch and defined it as the outlet for the upper basin.

10. Select the *Frame* macro
11. Select the *Zoom* tool
12. Create a zoom box around the region identified by a box in Figure 4-4

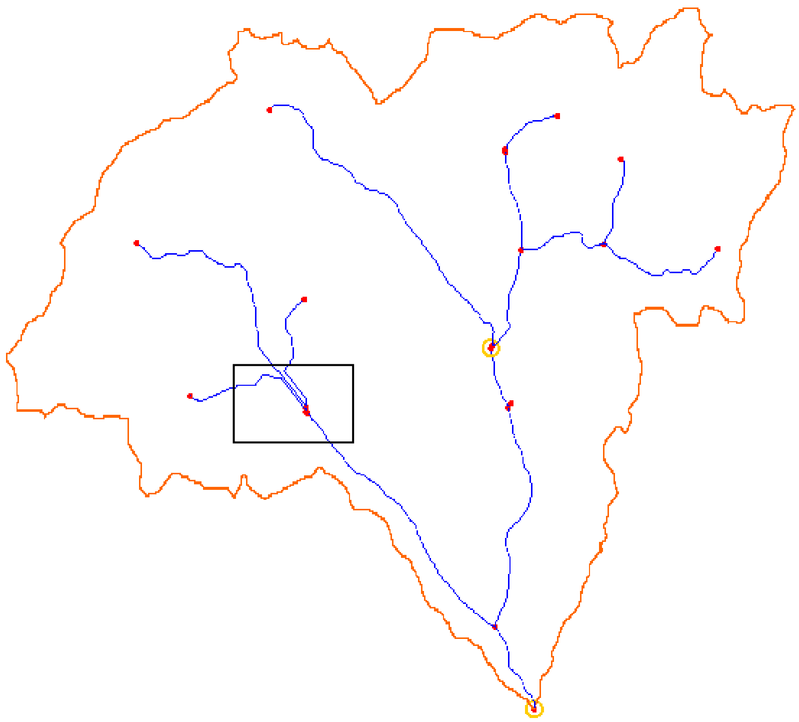
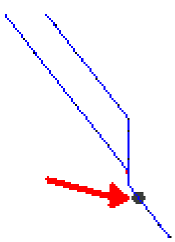


Figure 4-4: Zoom in on the area indicated by the rectangle



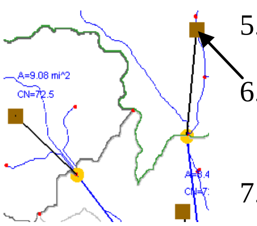
13. Select the *Select Feature Vertex* tool
14. Select the vertex that is just below the feature node where the streams branch
15. Select ***DEM* | *Node <-> Outlet***
16. Select the *Frame* macro
17. Select ***DEM* | *Delineate Basins Wizard***
18. Select *OK* to delete and recreate feature data
19. Select *OK*

20. Select *OK*

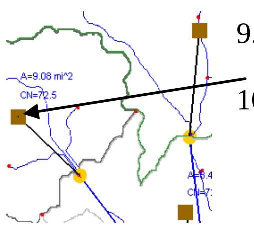
4.5.2 Updating the Basin Parameters

You will have to recompute the CN values and define precipitation and lag time for the basins.

1. Select the *Hydrologic Modeling* module
2. Select ***Calculators | Compute GIS Attributes***
3. Select *OK* and the CN values will be updated for all basins (they are actually very similar in this case because of the dominant soil polygon that covers the watershed)
4. Close the Curve Number Report



5. Select the *Select Basin* tool
6. Double-click on the upper right basin icon to bring up the Edit HEC-1 Parameters dialog
7. Select the *Basin Data* button
8. Change the name to *Right*



9. Select *OK*
10. Move the Edit HEC-1 Parameters dialog out of the way, if necessary, and click on the upper left basin icon to edit parameters for the upper left basin. Alternatively, you could select *Done* and then double-click on a basin to obtain the Edit HEC-1 Parameters dialog.
11. Select the *Basin Data* button
12. Change the name to *Left*
13. Select *OK*
14. Move the Edit HEC-1 Parameters dialog out of the way, if necessary, and click on the lower basin icon to edit parameters for the lower basin
15. Select the *Basin Data* button
16. Change the name to *CCTrib*
17. Select *OK*
18. Select *Done*

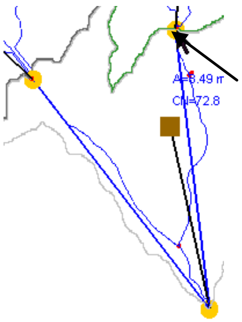
19. Select ***Edit | Select All*** to select all basins
20. Select ***HEC-1 | Edit Parameters*** to edit parameters for all basins at once
21. Select the *Precipitation* button
22. Select the *Basin Average* option
23. Set the Average Precipitation to be **1.8** in
24. Select the *Define Series* button
25. Choose the *typeII-24hour* curve in the Selected Curve drop down list
26. Select *OK*
27. Select *OK*
28. Select the *Unit Hydrograph Method* button
29. Make sure the *SCS dimensionless* option is chosen (it is the default)
30. Select the *Compute Parameters - Basin Data* button
31. Select *CCTrib* in the Basin window so that it is highlighted
32. Select the method to be *SCS Method* (near the bottom of the list)
33. Select *Right* in the Basin window so that it is highlighted
34. Select the Method to be *SCS Method* (near the bottom of the list)
35. Select *Left* in the Basin window so that it is highlighted
36. Select the method to be *SCS Method* (near the bottom of the list)
37. Select *OK*
38. Select *OK*
39. Select *Done*

4.5.3 Setting up the Routing Parameters

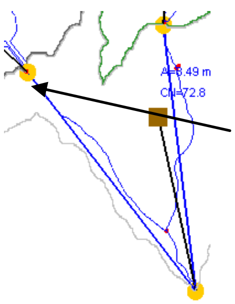
If you were to run HEC-1 now (you can if you want), you would see that the hydrographs from the upper basins would be combined with the lower basin hydrograph at the watershed outlet without any lag or attenuation because you have not yet set the routing parameters. You will now define a routing method,

which will instruct HEC-1 to compute lag and attenuation on the upper basin hydrographs before adding them to the lower hydrograph.

Routing for a reach is always defined at the upstream outlet of the reach in WMS.



1. Select the *Select Outlet* tool 
2. Double-click on the outlet (the yellow circle icon) of the upper right basin
3. Select the *Routing Data* button
4. Select the *Muskingum-Cunge* method for routing
5. Set the width (WD) field to be **5** (five feet wide)
6. Set the side slope value (Z) to be **1** (1:1 side slope)
7. Set the Manning's roughness (N) to be **0.05** (this is fairly rough, but we want to exaggerate the routing effects for this exercise)



8. Select *OK*
9. Select *Done*
10. Double-click on the outlet of the upper left basin
11. Select the *Routing Data* button
12. Select the *Muskingum-Cunge* method for routing
13. Set the width (WD) field to be **5**
14. Set the side slope value (Z) to be **1**
15. Set the Manning's roughness (N) to be **0.05**
16. Select *OK*
17. Select *Done*

4.5.4 Running HEC-1

You now have everything defined to run a three basin HEC-1 analysis that includes routing the upper basins through the reaches connecting them to the watershed outlet.

1. Select ***HEC-1 | Run Simulation***

2. Click the browse button next to the Input File

3. For the file name enter "*Routing*" and click *Save* (this specifies the file name but does not actually save it)

4. Verify that the *Save file before run* is toggled on

5. Select *OK*

6. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

7. While holding the SHIFT key down, select all of the hydrograph icons, double-clicking on the last one so that all hydrographs are drawn in the same plot window

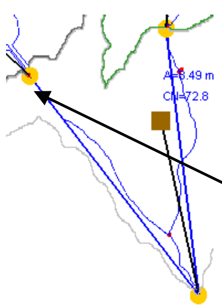
8. Close the plot window by selecting the X in the upper right corner

4.6 Modeling a Reservoir in HEC-1

There is an existing small reservoir at the outlet of the upper left basin. It has a storage capacity of 1000 ac-ft at the spillway level and 1540 ac-ft at the dam crest.

4.6.1 Defining a Reservoir in Combination with Routing

One of the routing methods available in HEC-1 is Storage routing, which can be used to define reservoir routing. However, in this case we are already using Muskingum-Cunge routing to move the hydrograph through the reach connecting the upper left basin to the watershed outlet so we must define the outlet as a reservoir so that we can route the hydrograph through the reservoir before routing it downstream.



1. Select the *Select Outlet* tool

2. Select the outlet of the upper left basin

3. Right-click on the outlet you have just selected and select ***Add | Reservoir***

4.6.2 Setting up the Reservoir Routing Parameters

In order to define reservoir routing with HEC-1 you must define elevation vs. storage (storage capacity curve) and elevation vs. discharge rating curves. You can enter values directly, or enter hydraulic structures and compute the values, but in this exercise you will enter the values directly. You will use the same

elevation values for both curves (this is common, but not a requirement in HEC-1).

For this example we want to have no outflow until the elevation in the reservoir reaches the spillway. Since HEC-1 linearly interpolates between consecutive points on the elevation-discharge and elevation-volume curves we will "trick" it by entering two points on the curves at essentially the same elevation (6821.99 ft and 6822 ft) with the first having no outflow and the second having the discharge over the spillway (640 cfs) as defined for this dam.

1. Double-click on the reservoir outlet point (it is now represented as a triangle since you have defined a reservoir at this location)

2. Select the *Reservoir Data* button

3. Change the Reservoir name to *Tcreek*

4. Set the Type of storage routing to *Reservoir*

5. Select the *Define* button to the right of the reservoir option

6. On the right side of this dialog you will define the Volume or storage capacity data. Choose the *Known Volume* option.

7. Toggle on the check boxes for *SV* (Volumes) and *SE* (Elevations)

8. Select the *Define* button to the right of the SV option

You will define separate XY series for Volumes, Elevations, and Discharges using the XY Series editor.

9. Select *New*

10. Change the name of the new curve to *"Volume"*

11. In the first seven edit fields enter the values **0**, **200**, **410**, **650**, **1000**, **1000**, **1540** (acre-ft of volume), as shown in Figure 4-5

12. Select the 8^{th} through 20^{th} edit fields and select the DELETE key so that the values are blank rather than zero. You can select them all at once (the way you do in a spreadsheet since this dialog behaves like a spreadsheet) by clicking in the top and while holding the mouse button down dragging to the last, or you can select one at a time.

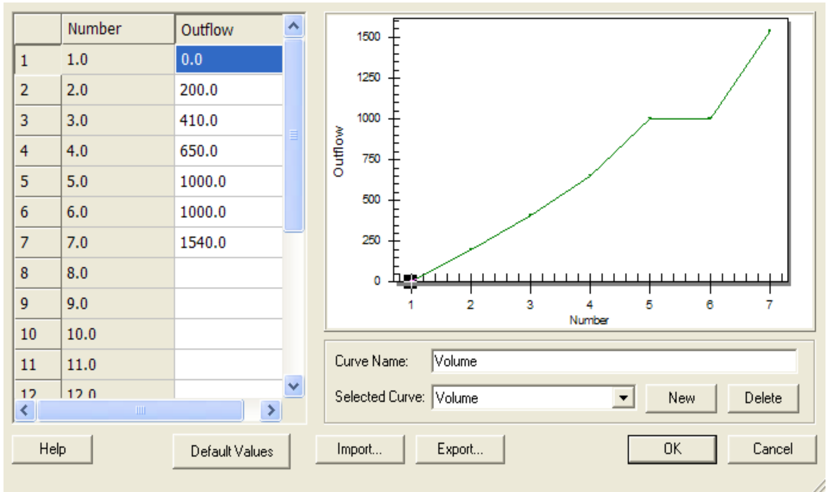


Figure 4-5: The XY Series editor for inputting volumes.

13. Select *OK*

14. Select the *Define* button to the right of the SE option

15. Select *New*

16. Change the name of the new curve to *"Elevation"*

17. In the first seven entry fields enter the following values: **6803**, **6808**, **6813**, **6818**, **6821.99**, **6822**, **6825** (feet of elevation)

18. Set the 8th through 20th fields blank instead of zero as with the volume series

19. Select *OK*

20. On the left side of this dialog you will define the Outflow or elevation - discharge data. Choose the *Known Outflow* option.

21. Toggle on the check boxes for *SQ* (Discharges) and *SE* (Elevations)

22. Select the *Define* button to the right of the SQ option

23. Select *New*

24. Change the name of the new curve to *"Discharge"*

25. In the first seven entry fields enter the following values: **0**, **0**, **0**, **0**, **640**, **640**, **7000** (cubic feet per second of flow). There is no outflow until the water reaches the spillway.

26. Set the 8th through 20th fields blank instead of zero as with the volume series

27. Select *OK*

28. Select the *Define* button to the right of the SE option

This time rather than creating a new curve you will select the elevation curve previously defined for the storage capacity curve.

29. Select the *Elevation* curve from the Selected Curve drop down list

30. Select *OK*

If you would like you may plot either the elevation-discharge or the elevation-volume curves by selecting the *Plot SQ-SE* or *Plot SV-SE* buttons. This will bring the curve into a plot window that you can export, print, or control the same way you can a hydrograph or any other plot in a plot window.

31. Select *OK*

The last thing you need to input to define reservoir routing is the initial conditions of the reservoir. The initial condition can be defined as an elevation, a discharge, or a volume (with the data you just entered HEC-1 can determine the initial condition of the other two based on the one you enter). For this example we will set the initial condition to an elevation four feet below the top of the spillway (the spillway corresponds to elevation 6822).

32. Under the Initial Condition Type select the *ELEV* option

33. Set the RSVRIC (reservoir initial condition) to be **6818**

34. Select *OK*

35. Select *Done*

4.6.3 Running HEC-1

You are now ready to save and run the HEC-1 file with the defined reservoir.

1. Select ***HEC-1 | Run Simulation***

2. Click the browse button  next to the Input File

3. For the file name enter "Reservoir" and click *Save* (this specifies the file name but does not actually save it)

4. Verify that the *Save file before run* is toggled on

5. Select *OK*

6. Ensure that *Read solution on exit* is selected and select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

7. After HEC-1 runs you can open any (or multiple within the same plot window by holding down the SHIFT key to multi-select) of the hydrographs by double-clicking on the corresponding icon.

8. Close all plot windows before moving on

4.7 Reviewing Output

It should be emphasized here that while WMS makes it easy to set up a HEC-1 model and compute a result, it is not a substitute for understanding the basic theory and equations used in HEC-1. You are encouraged to read the HEC-1 manual found in the documents directory distributed with WMS and other texts on hydrologic modeling. You are also encouraged to review the HEC-1 output file that is generated with each simulation in order to glean more understanding about how your model is working.

1. Select ***File | Edit File***

2. Find and open the file named *"reservoir.out"*

3. Select *OK* to open the file with Notepad

4. Scroll through this file and examine what information HEC-1 saves to the output file. If you have errors running HEC-1 simulations you may often find the answer to the problem within the *.out output file.

4.8 Conclusion

This concludes the exercise defining HEC-1 files and displaying hydrographs. The concepts learned include the following:

- Entering job control parameters
- Defining basin parameters such as loss rates, precipitation, and hydrograph methodology a watershed analysis
- Defining routing parameters
- Routing a hydrograph through a reservoir

- Saving HEC-1 input files
- Reading hydrograph results

CHAPTER 5

HEC-HMS Interface

WMS has a graphical interface to HEC-HMS as well and this tutorial is similar to the previous except that it focuses on the use of the HMS interface. Geometric attributes such as areas, lengths, and slopes are computed automatically from the digital watershed. Parameters such as loss rates, base flow, unit hydrograph method, and routing data are entered through a series of interactive dialog boxes. Once the parameters needed to define an HMS model have been entered, an input file with the proper format for HMS can be written automatically. Since only parts of the HMS input file are defined in this chapter, you are encouraged to explore the different available options of each dialog, being sure to select the given method and values before exiting the dialog. Unlike HEC-1, you will need to export the HMS files from WMS and then run the HMS graphical user interface to view the results. In order to do this you should have the most recent version of HMS installed.

5.1 Objectives

As a review, you will delineate a watershed from a DEM. You will then develop a simple, single basin model using the delineated watershed to derive many of the parameters. Land use and soil shapefiles (downloaded from the Internet) will be used to develop a SCS curve number (CN) value. After establishing the initial HMS model, other variations will be developed, including defining multiple basins with reach routing and including a reservoir with storage routing.

5.2 Delineating the Watershed

Since the land use, soil type, and DEM data for our watershed all originate in the Geographic coordinate system, we will begin by opening them together and converting them to UTM coordinates. The land use and soil type data were downloaded from the Environmental Protection Agency (EPA) website. The DEM data used for this watershed were previously downloaded from the National Elevation Dataset website as was demonstrated in the DEM Basics exercise (Volume 1, chapter 4).

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open*** 
4. Locate the folder ***C:\Program Files\WMS81\tutorial\hec-1***
5. Select *NED GRIDFLOAT Header (*.hdr)* from the Files of type list of file filters
6. Open *“67845267.hdr”*
7. Select *OK*
8. When prompted if you want to convert the current coordinates select *No*

5.2.1 Create Land Use and Soil Coverages

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the coverage type to *Land Use*
4. Select *OK*
5. Create a new coverage once again and set its coverage type to *Soil Type*

5.2.2 Open the Soils Data

1. Make sure the *Soil Type* coverage is active in the Project Explorer
2. Right-click on *GIS Layers* in the Project Explorer and select ***Add Shapefile Data***

3. Open *"statsgo.shp"*

4. Right-click on *statsgo.shp* layer in the Project Explorer

5. Select ***Open Attribute Table***

Notice that the table has three fields named AREA, PERIMETER, and MUID

6. Select *OK*

5.2.3 Join Soils Database File Table to Shapefile Table

1. Right-click on *statsgo.shp* in the Project Explorer

2. Select ***Join Table to Layer***

3. Open *"statsgoc.dbf"*

4. Ensure that Shapefile Join Field and Table Join Field are both set to *MUID*

5. Change the Table Data Field to *HYDGRP*

6. Select *OK*

7. Right-click on *statsgo.shp* in the Project Explorer

8. Select ***Open Attribute Table***

Notice that the HYDGRP field is now a part of the shapefile.

9. Select *OK*

5.2.4 Convert Soil Shapefile Data to Feature Objects

1. Choose the *Select Shapes* tool 

2. Draw a selection box around the DEM extents

3. Select ***Mapping | Shapes -> Feature Objects***

4. Select *Next*

This window shows all of the attribute fields in the soils shape file. Because this file was derived from a standard NRCS statsgo file you will notice that the hydrologic soil groups field is named HYDGRP and so WMS will automatically map this to be the soil type. If the attribute field were named

anything other than HYDGRP then you would have to manually map it using the drop down list in the spreadsheet.

5. Make sure the HYDGRP field is mapped to the SCS soil type attribute
6. Select *Next*
7. Select *Finish*
8. Clear the selected polygons by single-clicking somewhere beyond the extents of the shapefile polygons
9. Hide the *statsgo.shp* file by toggling off its check box in the Project Explorer

5.2.5 Open the Land Use Data

1. Select the *Land Use* coverage in the Project Explorer to designate it as the active coverage
2. Right-click on GIS Layers in the Project Explorer and select ***Add Shapefile Data***
3. Open *"l_richut.shp"*
4. Choose the *Select Shapes* tool 
5. Draw a selection box around the DEM extents
6. Select ***Mapping | Shapes -> Feature Objects***
7. Select *Next*
8. Make sure the LUCODE field is mapped to the Land use attribute
9. Select *Next*
10. Select *Finish*
11. Hide the *l_richut.shp* file by toggling off its check box in the Project Explorer

5.2.6 Convert/Set the Coordinate System of the Data

1. Select ***Edit | Coordinate Conversion***
2. Select the *Specify* option in the *Current Projection* section of the *Reproject Current* dialog

3. Select *Set Projection*

4. Set *Projection* to *Geographic*, and *Datum* to *NAD 83*

5. Select *OK*

6. Set Vertical Units to *Meters*

7. In the *New Projection* section select *Global Projection*

8. Select *Set Projection*

9. Set *Projection* to *UTM*, *Datum* to *NAD 83*, *Planar Units* to *METERS*, and *Zone* to *12 (114°W - 108°W – Northern Hemisphere)*

10. Select *OK*

11. Set vertical units to *Meters*

12. Select *OK* to convert the data

13. Since we will not be using them until later, hide the *Land Use* and *Soil Type* coverages by toggling off their check boxes in the Project Explorer

14. Select the *Drainage* coverage from the Project Explorer to make sure it is the active coverage

5.2.7 Delineate the Watershed

1. Select the *Drainage* module 

2. Select the *Frame* macro 

3. Select ***DEM | Compute TOPAZ Flow Data***

4. Select *OK*

5. Select *OK* in the Units dialog

6. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

7. Select the *Zoom* tool 

8. Zoom in by dragging a box as illustrated in Figure 4-1

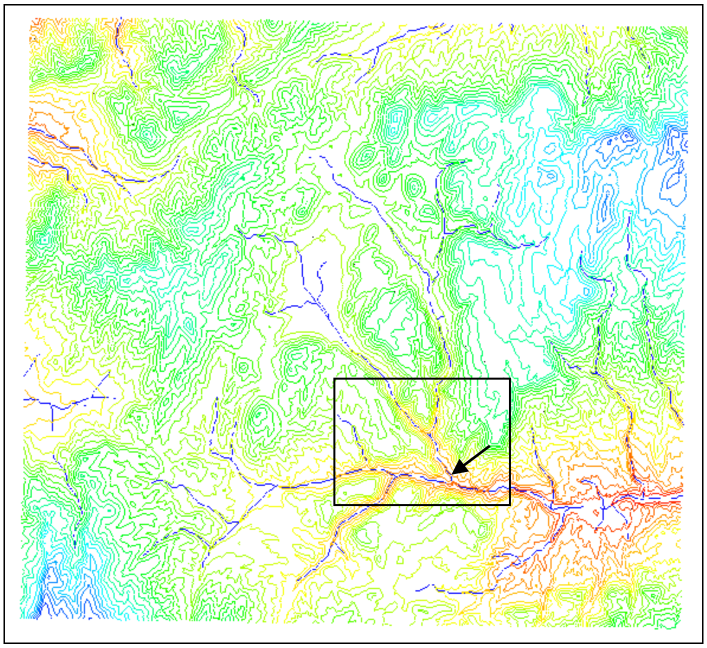


Figure 5-1: Zoom in on the area bounded by the rectangle above

9. Select the *Create Outlet Point* tool

10. Create a new outlet point where the tributary you just zoomed in on separates from the main stream as illustrated by the arrow in Figure 4-1. Make certain that the outlet point is on the tributary and not part of the main stream. Also, the outlet needs to be inside one of the flow accumulation (blue) cells. WMS will move the outlet to the nearest flow accumulation cell if you do not click right in one of the flow accumulations cells.

11. Select the *Frame* macro

12. Select ***DEM | Delineate Basins Wizard***

13. Select *OK*

14. Select *OK*

You have now completed the delineation of a single watershed. In order to make the view clearer for defining the hydrologic model you can turn off many of the DEM and other display options.

15. Right-click on *DEM* in the Project Explorer and select ***Display Options***

16. On the DEM tab, toggle off the display for *Watershed*, *Stream*, *Flow Accumulation*, and *DEM Contours*

17. On the Map tab, toggle *Vertices* off
18. Select *OK*

5.3 Single Basin Analysis

The first simulation will be defined for a single basin. You will need to enter the global, or Job Control parameters as well as basin and meteorological data.

5.3.1 Setting up the Job Control

Most of the parameters required for a HEC-HMS model are defined for basins, outlets, and reaches. However, there are some “global” parameters that control the overall simulation and are not specific to any basin or reach in the model. These parameters are defined in the WMS interface using the Job Control dialog.

1. Switch to the *Hydrologic Modeling* module 
2. HEC-1 should be the default model, so change the default model to HEC-HMS by selecting it from the drop down list of models found in the Edit Window
3. Select ***HEC-HMS | Job Control***
4. Enter *“Clear Creek Tributary”* in the Name: field
5. In the Description: field you can enter Your name

By default the simulation is set to run for 24 hours starting from today’s date at 15 minute intervals. We want to run this simulation for 25 hours at five minute intervals.

6. Add one hour to the Ending time
7. Change the Time interval to *5 Minutes*
8. Select the Basin Options tab
9. Enter *“Clear Creek Tributary”* in the Name: field
10. Set the Basin Model Units to *US customary (English),* which should already be the default

Setting the computation units DOES NOT cause any units conversion to take place. You are simply telling HEC-1 that you will provide input units in English units (sq. miles for area, inches for rain, feet/miles for length) and

expect results of computation to be in English units (cfs). If you specify Metric then you must ensure that input units are metric (sq. kilometers, mm for rain, meters/kilometers for length) and results will be in metric (cms).

11. Select the Meteorological Options tab
12. Enter *"Clear Creek Tributary"* in the Name: field

You will note that HEC-HMS includes advanced options for long term simulation and local inflows at junctions, but we will not explore these options in this model.

13. Select *OK*

5.3.2 Setting up the Meteorological Data

In HEC-1 precipitation is handled as a Basin Data attribute, however for HEC-HMS precipitation is defined separately in the Meteorological Data. This is because of the ability of HEC-HMS to model long term simulations that require additional information and often a lot more input.

1. Select ***HEC-HMS | Meteorological Parameters***
2. Set the Precipitation Method to *SCS Hypothetical Storm*
3. Set the Storm Selection to *Type II*
4. Set the Storm Depth to *1.8* (inches)
5. Select *OK*

5.3.3 Setting up the Basin Data Parameters

In the first simulation you will treat the entire watershed as a single basin.

1. Select the *Select Basin* tool 
2. Double-click on the brown basin icon labeled 1B. Double-clicking on a basin or outlet icon always brings up the parameter editor dialog for the current model (in this case HEC-HMS)
3. Notice that the area has been calculated (in this case in sq. miles because we are performing calculations in English units).
4. Change the name to *CCTrib*
5. Enter "*Main Branch*" in the description.

Displaying and Showing options allows you to see only those variables for which you wish to enter data. For example in this case toggling on the Loss Rate Method allows you to pick which method you want to use (in this case the method we want is the default). You then toggle the display for the different parameters associated with a given methodology from the show column. In our case we can now see in the Properties window the Loss Rate Method and the parameters for the SCS Curve Number method.

The HMS-Properties window is versatile in that it allows you to see properties for all or selected basins, junctions, reaches, reservoirs, etc.

6. Toggle on the Display of the *Loss Rate Method* option

7. Toggle the *SCS Curve Number* from the Show column in the Display options window

8. Enter an SCS Curve Number of **70**. We will compute a CN value from actual land use and soil files later.

For the SCS CN method initials losses are estimated as 20% of the maximum storage value computed from the CN when the initial loss is zero. If you wish to override this computation then you would enter a value other than zero. For now we will assume there is no impervious area.

9. Toggle on the Display of the *Transform* option (you may have to scroll vertically in the Display options window)

10. Show the *SCS* parameters by toggling this option on in the Display options window)

11. Scroll horizontally in the Properties window and choose the *Compute* button under Basin Data (the SCS transform method is the default)

12. Set the Computation Type to *Compute Lag Time* (the default)

13. Set the Method drop down list to *SCS Method* (near the bottom of the list)

14. Select *OK* to update the computed lag time for the SCS dimensionless method (scroll horizontally to view if you would like)

15. Select *OK*

You now have all of the parameters set to run a single basin analysis.

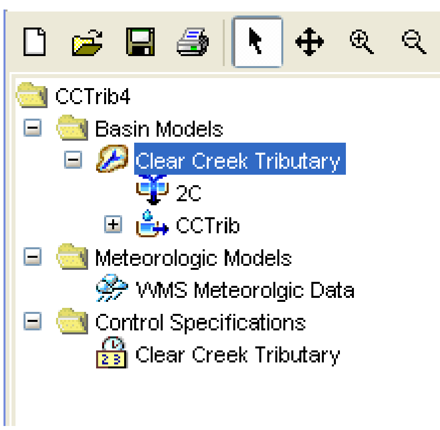
5.3.4 Running HEC-HMS

Whenever you run an HEC-HMS simulation, you must save the information created in WMS to HEC-HMS files and then load it as a project in HEC-HMS.

This tutorial is not a comprehensive review of HEC-HMS but should give you an idea of how to open a project created by WMS, run an analysis and view some basic results.

1. Right click on *Drainage Coverage Tree* in the Project Explorer and select ***Save HMS File***
2. Change the HMS project file to *CCTrib*
3. Start HEC-HMS on your computer
4. Select ***File |Open***
5. Select the *Browse* button and browse to the location where you just saved your HMS Project from WMS (by default this will be in the hec-1 directory of your tutorial files)
6. Select the *CCTrib.hms* project file
7. From the HEC-HMS project explorer expand the Basin Models, Meteorologic Models and Control Specifications folders
8. Expand the Clear Creek Tributary basin model and then select it

The basin model map should appear and your project explorer should look something like the picture below.



9. Select ***Compute | Create Simulation Run***
10. Change the Run Name to *CCTrib 1*
11. Click *Next*, *Next*, *Next* and *Finish* to set up the simulation run
12. Select ***Compute | Select Run | Select Run -> CCTrib 1***

13. Select ***Compute | Compute Run [CCTrib 1]*** or the Compute Current Run macro 

14. When finished computing select *Close*

15. Select the CCTrib basin under the Clear Creek Tributary basin model from the HEC-HMS project explorer

16. Select ***Results | Global Summary Table*** and explore

17. Select ***Results | Element Graph*** and explore

18. Select ***Results | Element Summary Table*** and explore

19. Select ***Results | Element Time-Series Table*** and explore

You now have a completed HEC-HMS simulation for a single basin and the resulting hydrograph for the CCTrib subbasin element should look something like the one shown in Figure 5-2.

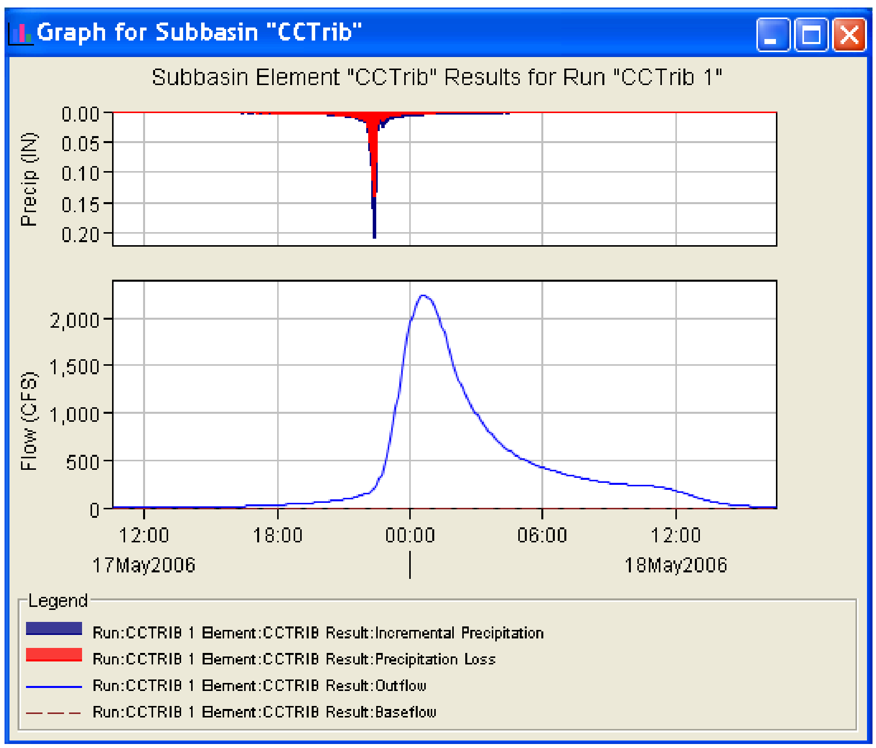


Figure 5-2: Solution hydrograph for HEC-HMS simulation

20. You may continue to explore the HEC-HMS input parameters passed from WMS and computed results or any other options

21. When finished close the Global and Element summary tables and graph windows and exit HEC-HMS by selecting ***File | Exit***

22. Select *Yes* when prompted to save the project.

5.4 Computing the CN Using Land Use and Soils Data

In the initial simulation you estimated a CN, but with access to the Internet it is simple to compute a composite CN based on digital land use and soils files. This was demonstrated in more detail in the Advanced Feature Objects exercise (Volume 1, chapter 6), but you will go through the steps here as a review.

5.4.1 Computing a Composite CN

At the beginning of this tutorial you loaded digital land use and soils files for the purpose of calculating a CN. In addition to this data, you must have a table defined that relates CN values for each of the four different hydrologic soil groups (A, B, C, D) for each land use. This is described in detail at the gsda website (http://www.xmswiki.com/wiki/GSDA:GSDA), and in the Advanced Feature Objects exercise (Volume 1, chapter 6). For this exercise you will read in an existing file (you can examine it in a text editor if you wish) and compute the CN numbers.

1. While it is not necessary to have the *Land Use* and *Soil Type* coverages displayed for the computations to work you may wish to make them visible again by toggling on their check boxes in the Project Explorer

2. Select the *Drainage* coverage to make sure it is the active coverage

3. Select the *Hydrologic Modeling* module 

4. Select ***Calculators | Compute GIS Attributes***

5. Select the *Import* button to load the mapping table

6. Select *OK* to overwrite the current definition

7. Find and open the file named *"scsland.tbl"*

8. Select *OK* to compute the CN from the land use and soils layers

You should see the computed CN displayed in the Runoff Curve Number Report and above the area label in the WMS graphics window.

9. Close the Runoff Curve Number Report

5.4.2 Running HEC-HMS

You can now run another simulation to compare the results with the modified CN value.

1. Right click on *Drainage Coverage Tree* in the Project Explorer and select ***Save HMS File***
2. Name the HMS project file *CCTribCN* and Save
3. Start HEC-HMS on your computer
4. Select ***File |Open***
5. Select the *Browse* button and browse to the location where you just saved your HMS Project from WMS (by default this will be in the hec-1 directory of your tutorial files)
6. Select the *CCTribCN.hms* project file
7. From the HEC-HMS project explorer expand the Basin Models, Meteorologic Models and Control Specifications folders
8. Expand the Clear Creek Tributary basin model and then select it
9. Select ***Compute | Create Simulation Run***
10. Change the Run Name to *CCTribCN 1*
11. Click *Next*, *Next*, *Next*, and *Finish* to set up the Run
12. Select ***Compute | Select Run | Select Run CCTribCN 1***
13. Select ***Compute | Compute Run [CCTribCN 1]*** or the Compute Current Run macro 
14. When finished computing select *Close*
15. Select the CCTrib basin under the Clear Creek Tributary basin model from the HEC-HMS project explorer
16. Select ***Results | Global Summary Table*** and explore
17. Select ***Results | Element Graph*** and explore
18. Select ***Results | Element Summary Table*** and explore
19. Select ***Results | Element Time-Series Table*** and explore

You may continue to explore the HEC-HMS input parameters passed from WMS and computed results or any other options

20. When finished close the Global and Element summary tables and graph windows and exit HEC-HMS by selecting ***File* | *Exit***

21. Select *Yes* when prompted to save the project.

5.5 Adding Sub-basins and Routing

You will now subdivide the watershed into two upper basins and one lower basin and define routing for the reaches that connect the upper basins to the watershed outlet.

5.5.1 Delineating the Sub-basin

1. Select the *Zoom* tool 

2. Create a zoom box around the region identified by a box in Figure 4-3

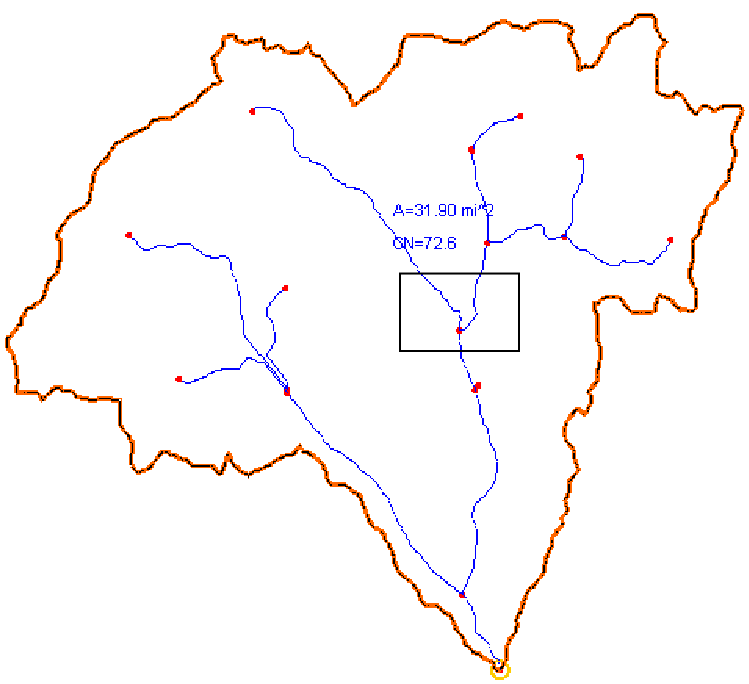


Figure 5-3: Zoom in on the area indicated by the rectangle.

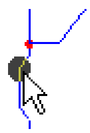
3. Right-click on the *Drainage* coverage in the Project Explorer and select ***Display Options*** 

4. On the Map tab toggle on *Vertices*

5. Select *OK*

6. Select the *Drainage* module

7. Select the *Select Feature Vertex* tool



8. Select the vertex that is just below the main branching point you just zoomed in around

9. Select ***DEM* | *Node* <-> *Outlet***

You created the outlet point just below the branch in order to have a single upstream basin. If you wanted a separate basin for each upstream branch you could define the branching node to be an outlet. Thus, WMS would automatically assume that you want separate basins for each branch, so we have created a node just downstream of the branch and defined it as the outlet for the upper basin.

10. Select the *Frame* macro

11. Select the *Zoom* tool

12. Create a zoom box around the region identified by a box in Figure 4-4

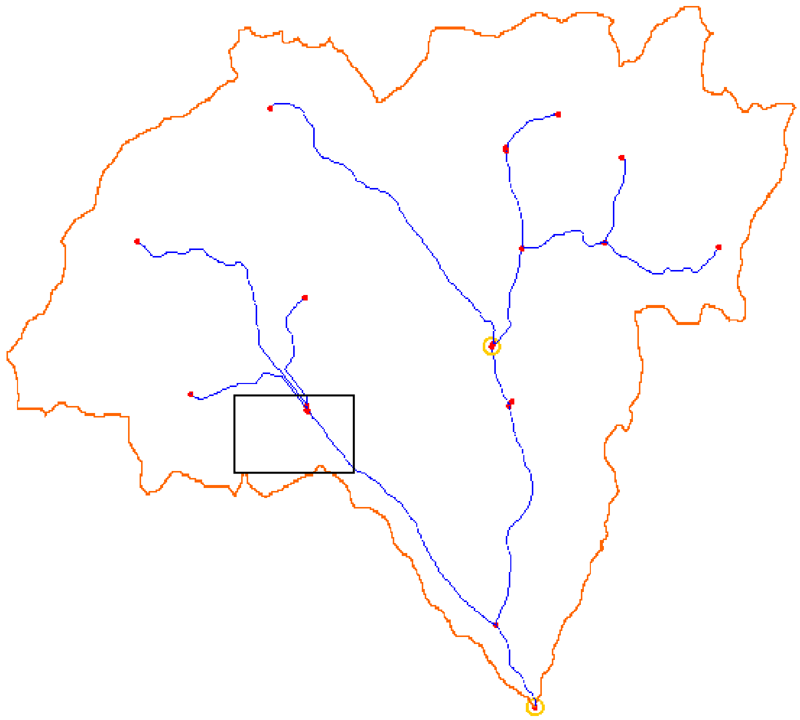
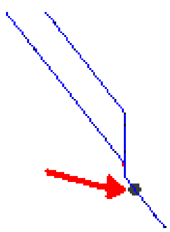
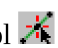


Figure 5-4: Zoom in on the area indicated by the rectangle



13. Select the *Select Feature Vertex* tool 

14. Select the vertex that is just below the feature node where the streams branch

15. Select ***DEM* | *Node*** **<->** ***Outlet***

16. Select the *Frame* macro 

17. Select ***DEM* | *Delineate Basins Wizard***

18. Select *OK* to delete and recreate feature data

19. Select *OK*

20. Select *OK*

5.5.2 Updating the Basin Parameters

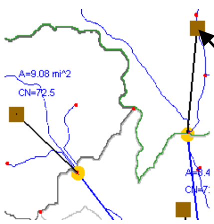
You will have to recompute the CN values and define precipitation and lag time for the basins.

1. Select the *Hydrologic Modeling* module 

2. Select ***Calculators | Compute GIS Attributes***

3. Select *OK* and the CN values will be updated for all basins (they are actually very similar in this case because of there is one dominant soil polygon that covers the watershed)

4. Close the Curve Number report



5. Select the *Select Basin* tool

6. Double-click on the upper right basin icon to bring up the HMS Properties dialog

7. Set the Show: option to *All*

8. Change the name of the upper right basin to *Right*

9. Change the name of the upper left basin to *Left*

10. Change the name of the lower basin to *CCTrib*

11. Toggle on the Display of the *Loss Rate Method* and Show *SCS Curve Number*

Because the CN values have been computed automatically you do not need to change anything here.

12. Toggle on the Display of *Transform* methods and Show *SCS*

13. For each basin choose the Define button under Compute Basin Data and define the equation and use the *Compute Lag Time* computation method with the *SCS Method*

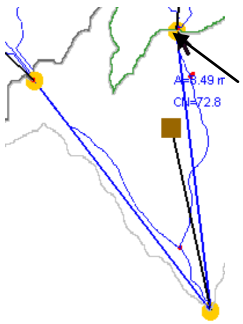
You should now have a computed lag time for each basin (all about 1 hour)

14. Select *OK*

5.5.3 Setting up the Routing Parameters

If you were to run HEC-HMS right now (you can if you want), you would see that the hydrographs from the upper basins would be combined with the lower basin hydrograph at the watershed outlet without any lag or attenuation because you have not yet set the routing parameters. You will now define a routing method, which will instruct HEC-HMS to compute lag and attenuation on the upper basin hydrographs before adding them to the lower hydrograph.

Routing for a reach is always defined at the upstream outlet of the reach in WMS.



1. Select the *Select Outlet* tool 
2. Double-click on the outlet (the yellow circle icon) of the upper right basin
3. Make sure the Type field at the top left of the dialog is set to *Reaches*
4. Set the Show option to *Selected* again
5. Toggle on the Display of *Method* and Show *Muskingum Cunge Std.*
6. Change the Routing Method to *Muskingum Cunge*.
7. Set the bottom width field to be **5** (five feet wide)
8. Set the side slope value to be **1** (1:1 side slope)
9. Set the Manning's roughness (N) to be **0.05** (this is fairly rough, but we want to exaggerate the routing effects for this exercise)

10. Select *OK*
11. Double-click on the outlet of the upper left basin
12. Make sure the Type field at the top left of the dialog is set to *Reaches*
13. Change the Routing Method to *Muskingum Cunge*.
14. Set the bottom width field to be **5** (five feet wide)
15. Set the side slope value to be **1** (1:1 side slope)
16. Set the Manning's roughness (N) to be **0.05** (this is fairly rough, but we want to exaggerate the routing effects for this exercise)
17. Select *OK*

5.5.4 Running HEC-HMS

You now have everything defined to run a three basin HEC-HMS analysis that includes routing the upper basins through the reaches connecting them to the watershed outlet.

1. Right click on *Drainage Coverage Tree* in the Project Explorer and select ***Save HMS File***
2. Name the HMS project file *CCTribRoute* and Save
3. Start HEC-HMS on your computer

4. Select ***File |Open***

5. Select the *Browse* button and browse to the location where you just saved your HMS Project from WMS (by default this will be in the hec-1 directory of your tutorial files)

6. Select the *CCTribRoute.hms* project file

7. From the HEC-HMS project explorer expand the Basin Models, Meteorologic Models and Control Specifications folders

8. Expand the Clear Creek Tributary basin model and then select it

9. Select ***Compute | Create Simulation Run***

10. Change the Run Name to *CCTribRoute 1*

11. Click *Next*, *Next*, *Next*, and *Finish* to set up the simulation run

12. Select ***Compute | Select Run | Select Run CCTribRoute 1***

13. Select ***Compute | Compute Run [CCTribRoute 1]*** or the Compute Current Run macro 

14. When finished computing select *Close*

15. Select different elements (basins, junctions, reaches) and view results

16. Select ***Results | Global Summary Table*** and explore

17. Select ***Results | Element Graph*** and explore

18. Select ***Results | Element Summary Table*** and explore

19. Select ***Results | Element Time-Series Table*** and explore

You may continue to explore the HEC-HMS input parameters passed from WMS and computed results or any other options

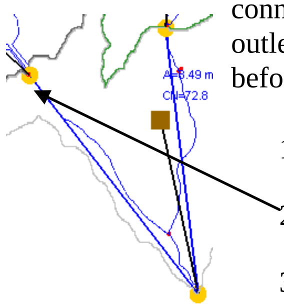
20. When finished close the Global and Element summary tables and graph windows and exit HEC-HMS by selecting ***File | Exit***

21. Select *Yes* when prompted to save the project.

5.6 Modeling a Reservoir in HEC-HMS

There is an existing small reservoir at the outlet of the upper left basin. It has a storage capacity of 1000 ac-ft at the spillway level and 1540 ac-ft at the dam crest.

5.6.1 Defining a Reservoir in Combination with Routing



One of the routing methods available in HEC-HMS is Storage routing, which can be used to define reservoir routing. However, in this case we are already using Muskingum-Cunge routing to move the hydrograph through the reach connecting the upper left basin to the watershed outlet so we must define the outlet as a reservoir so that we can route the hydrograph through the reservoir before routing it downstream.

1. Select the *Select Outlet* tool 
2. Select the outlet of the upper left basin
3. Right-click on the outlet and select ***Add | Reservoir***

5.6.2 Setting up the Reservoir Routing Parameters

In order to define reservoir routing with HEC-HMS you must define elevation vs. storage (storage capacity curve) and elevation vs. discharge rating curves. You can enter values directly, or enter hydraulic structures and compute the values, but in this exercise you will enter the values directly. You will use the same elevation values for both curves.

For this example we want to have no outflow until the elevation in the reservoir reaches the spillway. Since HEC-HMS linearly interpolates between consecutive points on the elevation-discharge and elevation-volume curves we will “trick” it by entering two points on the curves at essentially the same elevation (6821.99 ft and 6822 ft) with the first having no outflow and the second having the discharge over the spillway (640 cfs) as defined for this dam.

1. Double-click on the reservoir outlet point (it is now represented as a triangle since you have defined a reservoir at this location)
2. Change the Reservoir name to *Tcreek*
3. Set the Method drop down to be *Elevation-Storage-Discharge*

What you need to input to define reservoir routing is the initial conditions of the reservoir. The initial condition can be defined as an elevation, a discharge, or a volume. For this example we will set the initial condition to an elevation

four feet below the top of the spillway (the spillway corresponds to elevation 6822).

4. Set the Initial drop down to be *Elevation*

5. Enter *6818* for the Initial Value (this should be the default already)

6. Select the *Define Elevation-Storage* button

7. Select *New*

8. Change the name of the new curve to *"Elevation-Storage"*

9. In the first seven entry fields in the first column enter the following values: *6803, 6808, 6813, 6818, 6821.99, 6822, 6825* (feet of elevation)

10. In the first seven entry fields in the second column enter the following values: *0, 200, 410, 650, 999.99, 1000, 1540* (acre-feet of volume)

11. Select *OK*

12. Select the *Define Storage-Discharge* button

You will define separate XY series for Volumes, Elevations, and Discharges using the XY Series editor.

13. Select *New*

14. Change the name of the new curve to *"Storage-Discharge"*

15. In the first seven edit fields in the first column enter the values *0, 200, 410, 650, 999.99, 1000, 1540* (acre-ft of volume)

16. In the first seven entry fields in the second column enter the following values: *0, 0, 0, 0, 639.99, 640, 7000* (cubic feet per second of flow). There is no outflow until the water reaches the spillway.

17. Select *OK*

18. Select *OK*

5.6.3 Running HEC-HMS

You now have everything defined to run a three basin HEC-HMS analysis that includes routing the upper basins through the reaches connecting them to the watershed outlet.

1. Right click on *Drainage Coverage Tree* in the Project Explorer and select ***Save HMS File***

2. Name the HMS project file *CCTribReservoir* and Save

3. Start HEC-HMS on your computer

4. Select ***File |Open***

5. Select the *Browse* button and browse to the location where you just saved your HMS Project from WMS (by default this will be in the hec-1 directory of your tutorial files)

6. Select the *CCTribReservoir.hms* project file

7. From the HEC-HMS project explorer expand the Basin Models, Meteorologic Models and Control Specifications folders

8. Expand the Clear Creek Tributary basin model and then select it

9. Change the Run Name to *CCTribReservoir 1*

10. Click *Next*, *Next*, *Next*, and *Finish* to set up the simulation run

11. Select ***Compute | Create Simulation Run***

12. Select ***Compute | Select Run | Select Run CCTribReservoir 1***

13. Select ***Compute | Compute Run [CCTribReservoir 1]*** or the Compute Current Run macro 

14. When finished computing select *Close*

15. Select different elements (basins, junctions, reaches, reservoirs) and view results

16. Select ***Results | Global Summary Table*** and explore

17. Select ***Results | Element Graph*** and explore

18. Select ***Results | Element Summary Table*** and explore

19. Select ***Results | Element Time-Series Table*** and explore

You may continue to explore the HEC-HMS input parameters passed from WMS and computed results or any other options

20. When finished close the Global and Element summary tables and graph windows and exit HEC-HMS by selecting ***File | Exit***

21. Select *Yes* when prompted to save the project.

5.7 Conclusion

This concludes the exercise defining HEC-HMS files and displaying hydrographs. The concepts learned include the following:

- Entering job control parameters
- Defining basin parameters such as loss rates, precipitation, and hydrograph methodology a watershed analysis
- Defining routing parameters
- Routing a hydrograph through a reservoir
- Saving and running HEC-HMS simulations

CHAPTER 6

Rational Method Interface

The Rational Method is one of the simplest and best known methods routinely applied in urban hydrology. Peak flows are computed from the simple equation:

$$Q = kCiA$$

where:

Q -	Peak flow
k -	Conversion factor
C -	Runoff coefficient
i -	Rainfall intensity
A -	Area

In this exercise you will learn how to solve problems using a digital terrain model and the Rational Method.

6.1 Reading in Terrain Data

The terrain model used in this exercise is a TIN for a small portion of a city. The elevation data was obtained by digitizing a contour map.

1. Close all instances of WMS
2. Open WMS

3. Select ***File* | *Open***
4. Locate the folder ***C:\Program Files\WMS81\tutorial\rational***
5. Find and open *"afrational.h5"*
6. Select the *Drainage* module
7. Select ***DEM* | *Compute Basin Data***
8. The Model units should be feet. Set the Parameter units to be *Acres* for Basin Areas and *Feet* for Distances.
9. Select *OK*

6.2 Running a Rational Method Simulation

The areas computed from the TIN can now be used in setting up a Rational Method simulation of the urban development. Each of the outlet points represents an inlet to a storm drain.

1. Select the *Hydrologic Modeling* module
2. Select *Rational* to be the current model from the drop down list of models at the top of the screen
3. Double-click the basin icon for the basin labeled *"*Upper*"* in Figure 6-1

The Rational Method dialog should appear. The parameters shown in the dialog correspond to the basin that was selected.

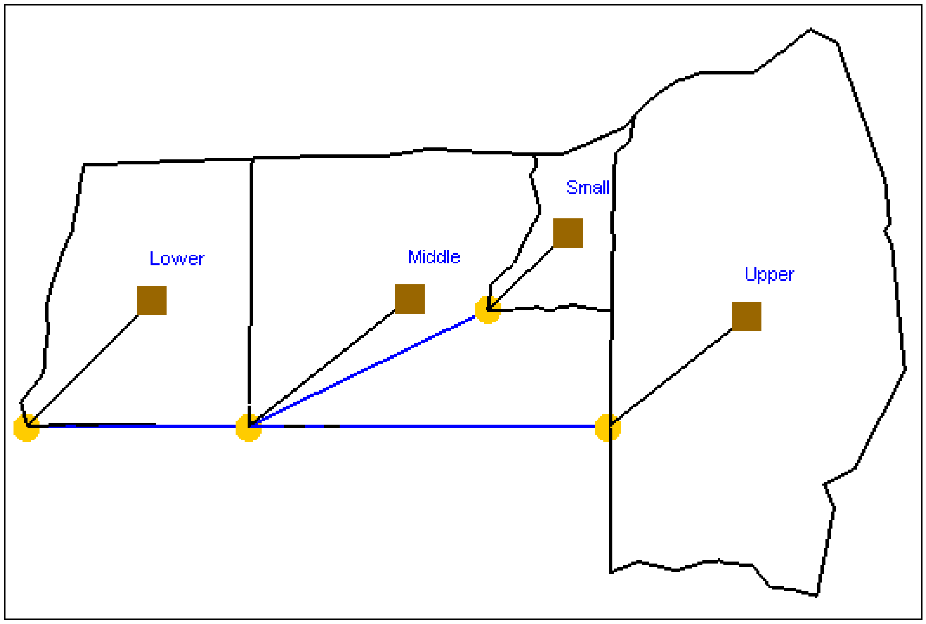


Figure 6-1: Basins.

6.2.1 Defining the Runoff Coefficients and Time of Concentration

The runoff coefficient, C, is used to account for losses between rainfall and runoff. The more developed a catchment is, the higher the C value it will have.

1. Enter a value of **0.20** for C
2. Enter a value of **22** for Time of Concentration
3. Click *OK*
4. Double click on the basin labeled "Small" in Figure 6-1
5. Enter a value of **0.35** for C
6. Enter a value of **6** for Time of Concentration
7. Repeat this process for the other two basins, using the table below to fill in values for C and t_c

Basin Name	Runoff Coefficient C	Time of Concentration t_c
Upper	0.20	22
Small	0.35	06
Middle	0.40	18
Lower	0.40	11

8. When you are finished entering the parameters choose *OK* on the Rational Method dialog

A runoff coefficient coverage could be used to automatically map C values and basin data, or a time computation coverage could be employed to determine Tc values, but they can also be computed/estimated separately and entered as demonstrated here.

6.2.2 Defining the Rainfall Intensity (i)

As part of the WMS interface to the Rational Method, you can compute IDF curves using either HYDRO-35, NOAA, or user defined data. For this exercise we will use HYDRO-35 data and a recurrence interval of 10 years.

1. Double-click the icon for the "Upper" catchment

2. Select the *Compute IDF Curves* button from the Basin section of the Rational Method dialog

3. Make sure the *HYDRO-35 Data (Eastern US)* radio group button is selected and select the *Define Storm Data* button

4. Enter the following values to define IDF curves using HYDRO-35

Time (min)	Depth (in)
2 yr. 5 min.	0.47
2 yr. 15 min.	0.97
2 yr. 60 min	1.72
100 yr. 5 min.	0.81
100 yr. 15 min.	1.75
100 yr. 60 min.	3.60

5. Select the *OK* button after correctly entering the rainfall values

The IDF curves for the 2, 5, 10, 25, 50, and 100 year recurrence intervals will be drawn, and values listed for selected times given in the windows on the right of the IDF Computation dialog.

6. From the text window in the upper right hand part of the dialog, click on the line of data for the 10-yr recurrence interval as shown in Figure 6-2

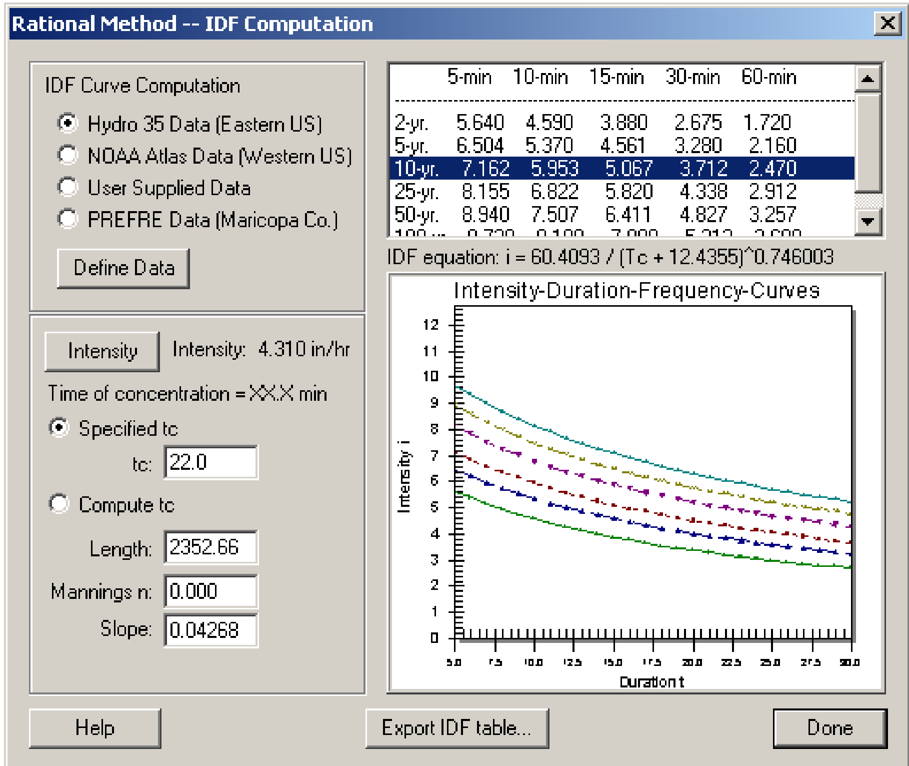


Figure 6-2: IDF Computation Dialog with 10-yr Recurrence Interval Selected.

The rainfall intensity is determined from the selected interval by using the previously defined value for time of concentration.

7. Compute i by selecting the *Compute Intensity* button

8. Select the *Done* button. The value of i computed using the IDF-Computation dialog will be placed in the edit field for this basin.

Note that the input for this basin is complete and a value for runoff Q has been computed.

The HYDRO-35 data only needs to be entered once (unless different data is to be used for different basins), so the rainfall intensity for the remaining basins can be defined using the following steps:

9. Double click on the basin icon for “Small”

10. Select the *Compute IDF Curves* button

11. Select the line of text for the 10-yr recurrence interval

12. Select the *Compute Intensity* button

13. Select *Done*

14. Repeat these steps for the “Middle” and “Lower” basins

15. When you are finished entering the parameters choose *Done* on the Rational Method dialog

6.2.3 Defining Hydrographs

As the data entry for each basin is completed, a peak flow (Q) is computed and listed in the text display window. The Rational Method equation does not produce a hydrograph. However, several different unit-dimensionless hydrographs can be used to distribute the peak flow through time to create a runoff hydrograph.

1. Double-click the basin labeled “Upper” in Figure 6-1

2. Select the *Compute Hydrographs* button

3. Select the *Rational method hydrograph* from the drop down list

4. Select *Done* to compute the hydrograph

5. Select *OK* on the Rational Method dialog

6. Double-click on the small hydrograph box to the upper right of the basin icon to open up a plot window of the hydrograph

You should see the hydrograph displayed in a plot window as shown in Figure 6-3.

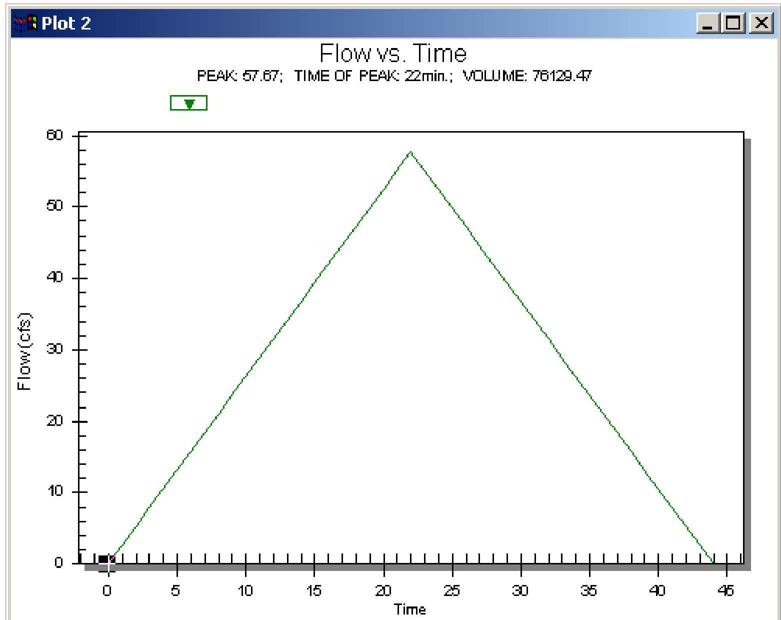


Figure 6-3 Rational Method Hydrograph for a Basin.

7. When you are finished viewing the hydrograph close the plot window by selecting the X in the upper right corner of the window

6.2.4 Defining Rainfall Intensities at the Outlet to Compute Runoff Hydrographs

WMS can determine composite rational method parameters at the outlet for computing hydrographs. The time of concentration at an outlet point is defined as the longest flow time from upstream basins (times of concentration) combined with any lag times from channels. The area is the cumulative upstream area, and the runoff coefficient is determined as an area weighted value from the upstream basins. With the time of concentration at the outlet defined you will need to determine the appropriate rainfall intensity. In order for WMS to compute peak flows and hydrographs at outlets you will need to define the travel time between outlets and the intensities for the time of concentration at each outlet.

1. Select the *Select Outlet* tool 

2. Double-click the outlet icon of “Upper” as shown in Figure 6-1 (be sure to select the circular outlet icon and not the square basin icon).

You will note in the Outlet portion of the Rational Method dialog that information upstream from this outlet has been aggregated (in the case though there is just one basin upstream). The longest flow time is listed for the time of concentration, a cumulative area, and a weighted C value.

3. Enter a value of **5** minutes for the Routing lag time
4. Select the *Compute IDF Curves* button in the Outlet column
5. Select the line of text for the 10-yr recurrence interval
6. Select the *Compute Intensity* button
7. Select *Done*
8. Select *OK*
9. Double-click the outlet icon of “Small”
10. Enter a value of **3** minutes for the Routing lag time
11. Select the *Compute IDF Curves* button in the Outlet column
12. Select the line of text for the 10-yr recurrence interval
13. Select the *Compute Intensity* button
14. Select *Done*
15. Select *OK*
16. Double-click on the next downstream outlet (the outlet icon of “Middle”)

Note that for this outlet the upstream areas are summed, the C values weighted, and the longest travel path computed from the upstream basin Tc’s and travel times between outlets determined.

17. Enter a value of **4** minutes for the Routing lag time
18. Select the *Compute IDF Curves* button in the Outlet column
19. Select the line of text for the 10-yr recurrence interval
20. Select the *Compute Intensity* button
21. Select *Done*
22. Select *OK*

The last or bottom-most outlet does not need to have a Routing lag time defined since the hydrograph accumulations will occur at this point, but you will still need to define the rainfall intensity.

23. Double-click on the bottom-most outlet point, “Lower”

24. Select the *Compute IDF Curves* button in the Outlet column

25. Select the line of text for the 10-yr-recurrence interval

26. Select the *Compute Intensity* button

27. Select *Done*

28. Select the *Compute Hydrographs* button in the Outlet column

29. Choose *Traditional method*

30. Choose the *Rational Method hydrograph* option

31. Select *Done*

32. Select *OK* in the Rational Method dialog

33. Double-click on the hydrograph icon for the most downstream outlet

34. Close the hydrograph plot window when you are done viewing by selecting the X in the upper right corner of the window

6.2.5 Combining Runoff from Multiple Basins

Besides the traditional method of computing peak flows and hydrographs for multiple sub-basins within a watershed, WMS will also allow you to lag hydrographs computed for basins and add at outlets in order to produce downstream peak flows and hydrographs.

1. Double-click on the downstream-most outlet point (outlet of the “Lower” basin)

2. Select the *Compute Hydrographs* button in the Outlet column

3. Choose the *Route by summing* method

4. Choose the *Rational method hydrograph* option

5. Select *Done*

6. Select *OK*

7. Double-click the hydrograph icon for the bottom-most outlet

You can now see the difference between these two methods as both hydrographs are plotted in the window.

8. Close the hydrograph plot window when you are done viewing by selecting the X in the upper right corner of the window

6.3 Adding a Detention Basin

If you compute runoff using the route by summing method then you can route hydrographs through detention basin structures defined at any of the outlet locations.

1. Double-click the outlet that defines the "Small" catchment in Figure 6-1

2. Select the *Define Reservoir* button in the Outlet column

You will now define a hypothetical detention basin for the "Small" catchment from approximate geometric parameters. WMS can compute a storage capacity curve for a rectangular basin. You could also enter a pre-computed storage capacity curve.

3. Click the *Define* button

4. Select the *Define Storage* button

5. Select the *Known Geometry* option

6. Enter **200** feet for Length

7. Enter **300** feet for Width

8. Enter a Depth of **30** feet

9. Enter a Side slope of **2**

10. Leave the Base elevation at **0.0** (it will be assumed on-grade at the outlet location)

11. Select *OK*

You will now define a standpipe and spillway (weir) for outlet structures and WMS will compute the elevation-discharge relationship automatically. In addition to standpipes and weirs you can define low-level outlets, or you can enter a pre-computed elevation-discharge relationship.

12. Select the *Define Discharges* button

13. Select the *Add Standpipe* button

14. Set the Pipe diameter to **4** feet

15. Set the Standpipe elevation to **15** feet

16. Select the *Add Weir* button

17. Set the Weir length to **20** feet

18. Set the Weir elevation to **25** feet

19. Select *OK* four times to return to the WMS window

You have now defined a detention facility that has a standpipe and a spillway for control structures. The incoming hydrograph to this outlet point will be routed through the detention facility before being routed downstream and combined with the hydrographs of other basins.

20. Select the bottom-most outlet point

21. Select the *Compute Hydrographs* button in the Outlet column

22. Choose the *Route by summing* method

23. Choose *Universal hydrograph method*

24. Select *Done*

25. Select *OK*

26. Double-click on the hydrograph box for the bottom-most outlet

6.4 Conclusion

In this exercise you have learned some of the options available for using the rational method in WMS. You will want to continue experimenting with the different options so that you can become familiar with all the capabilities in WMS for doing Rational Method simulations.

CHAPTER 7

National Streamflow Statistics Program (NSS) Interface

The National Streamflow Statistics program, developed by the USGS, provides a quick and easy way of estimating peak flows for ungaged watersheds. This data can be used in the design of culverts, flood-control structures, and flood-plain management. It utilizes regression equations that have been developed for each state. Most regression equations are functions of parameters such as area, slope, and runoff distance that are automatically computed by WMS when delineating a watershed.

You were already introduced to this program in a previous chapter discussing overlay and time of travel computations (Volume 2, Chapter 3). In this exercise you will have the chance to review data collection and starting a project from scratch. You will then run the NSS program for your selected area to compute the peak flows for the different return periods. If your equation ends up needing variables not derived from the DEM alone, then you might consider doing the general overlay in order to compute percentages of land use, soil, or rainfall for different regions.

7.1 Data Collection

You will begin by downloading the necessary files. Remember that you can go to the geospatial data acquisition (GSDA) web page on the xmswiki at http://xmswiki.com/wiki/GSDA:GSDA as a starting point.

1. Download a DEM from the National Elevation Dataset web site, or another similar site. This should be an area that contains a watershed of personal interest to you.

2. Open the DEM in a new instance of WMS and convert the coordinates as necessary. See the chapter on DEM basics if you need help (Volume 1, chapter 4).

3. Download the topographic map from the TerraServer by using the web services option in WMS if available, or by going to the TerraServer web site directly and downloading the image and accompanying world file. See the chapter on images if you need help (Volume 1, chapter 2).

4. Open the image and be sure that both DEM and image are in the same coordinate system

7.2 Basin Delineation

You will want to identify a watershed that is defined completely within your DEM. If your area does not contain a complete watershed, or the watershed you were trying to work on then you may need to repeat the delineation.

1. Switch to the *Drainage* module

2. Run TOPAZ and delineate your watershed

7.3 Running NSS

Now you should be prepared to run a basin NSS simulation.

1. Switch to the *Hydrologic Modeling* module

2. Make sure Model combo box is set to *NSS*

3. Choose the *Select Basin* tool

4. Double-click on your basin

5. Select your state and region

6. Define any variables not computed. You may wish to get land use, soils, or rainfall data and use it to automatically calculate other variables where needed.

7. Compute the peak flows

At this point you have your results and should feel confident that you could repeat this process again and again. You may wish to adjust display settings and add annotations in preparation of report documentation on a project.

When you are finished, do the following before moving on to the rest of the exercise.

8. Save the project if you wish
9. Select ***File | New***
10. Select *No* if asked if you want to save your changes

7.4 Utilizing an NSS Region Coverage

This portion of the exercise is optional. It will teach you how to create an NSS region coverage that can be used to map equations for a given state, using Florida as an example. If you do not have a need or interest for this you do not need to complete it.

The NSS Region coverage type allows WMS to automatically determine which regression equations to use for an NSS simulation. Additionally, if a drainage basin overlaps multiple NSS regions, the NSS Region coverage automates the calculations for the percentage of the watershed in each region.

You will now use an NSS Region coverage to automatically assign the region for an NSS simulation. This coverage was digitized from an image that displays the NSS regions of Florida. This image was obtained from the NSS documentation and you could make a similar map by scanning (or capturing if electronic) a map of the regions to create an image file, registering the image to a recognized coordinate system, digitizing the polygons (most states have less than 10 regions so it would take only a few minutes to digitize), and assigning the state and region. Details on how to scan images and create polygons by digitizing are given in the chapters on images and feature objects (see Volume 1). The USGS website (http://water.usgs.gov/software/NSS/) for NSS has images available in the state by state documentation of the equations that can be saved directly and then registered in WMS. In this example we will read in an image that has already been registered and polygons that were digitized from the image.

1. Switch to the *Map* module
2. Select ***File | Open***
3. Locate the folder ***C:\Program Files\WMS81\tutorial\nss***
4. Open *"NSSmap_FL.jpg"*

7.4.1 Assigning Regions to Feature Polygons

We see from the image that Florida has three regions: A, B, and C. You will now open a WMS MAP file that contains these three polygons in an NSS Region coverage. In the interest of time the polygons have already been digitized (see the chapter on feature objects in Volume 1 for more information about digitizing/building polygons), but the assignment of attributes (state/region) has been left for you to do.

1. Select ***File* | *Open***
2. Open *"NSSmap_FL.map"*

This file was digitized directly from the image. See Volume 1, Chapter 3: Basic Feature Objects for information on how to digitize features from images.

3. Choose the *Select Feature Polygon* tool
4. Double-click in the polygon corresponding to the region labeled C, as shown in Figure 7-1

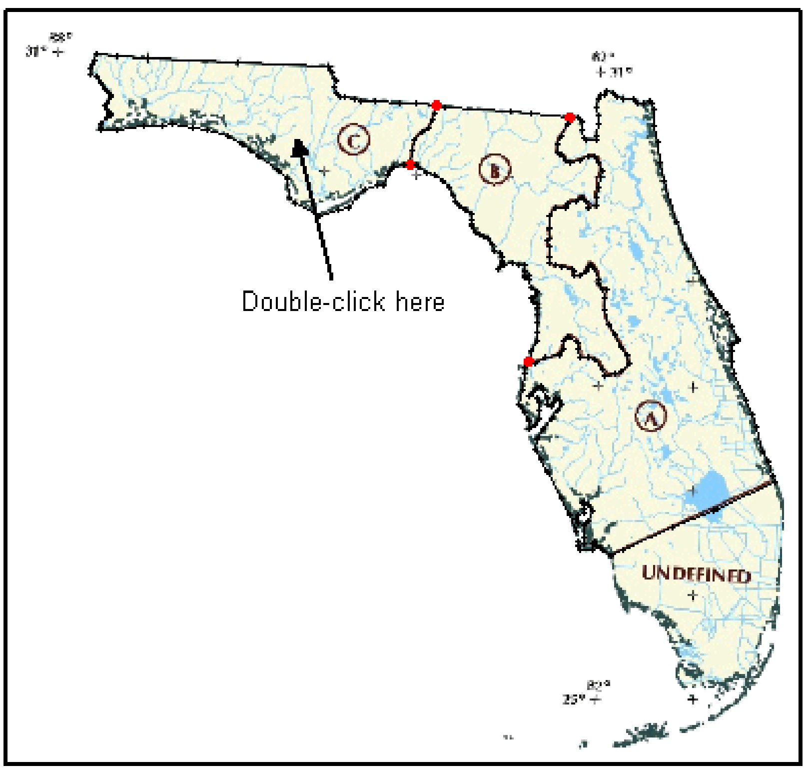


Figure 7-1: Image displaying NSS Regions for Florida

5. Choose *Florida* from the State list
6. Choose *Region C* from the NSS Region list

7. Select *OK*

8. Assign NSS Regions for the remaining two polygons in the same manner

7.4.2 Opening the Watershed

1. Select ***File | Open***

2. Open *"NSS_FL.wpr"*

At this point, the study area appears as a small polygon. You will zoom in to better distinguish the area.

3. Choose the *Zoom* tool

4. Zoom in on the region indicated in Figure 7-2

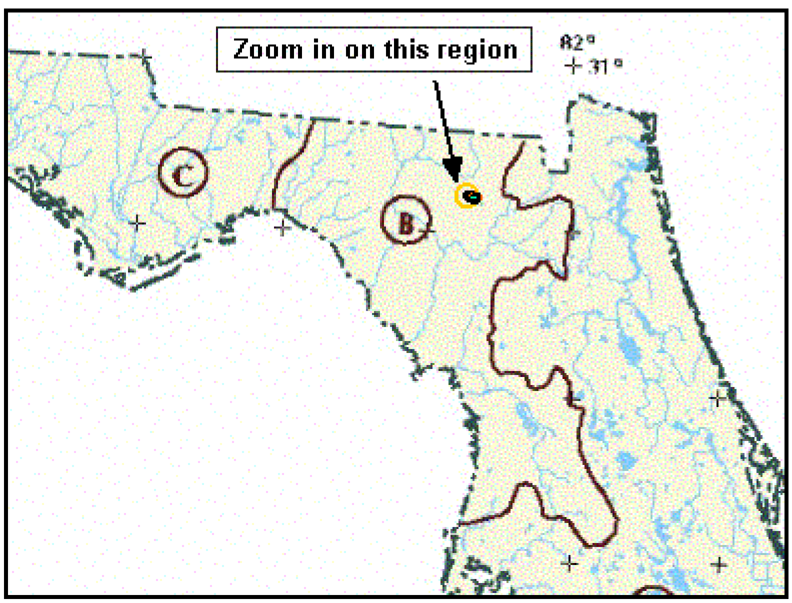


Figure 7-2: Zoom in on the watershed.

5. Right-click on *DEM* in the Project Explorer and select ***Display Options***

6. Choose *DEM Data*

7. Toggle off the check box for displaying *DEM Contours*

8. Select *OK*

9. Right-click on *Drainage* coverage and select ***Compute Basin Data***

10. Select the *Current Coordinates* button
11. Select *Global Projection*
12. Select *Set Projection*
13. Ensure *Meters* are selected in the Planar Units drop down box
14. Select OK
15. Ensure *Meters* are selected as the vertical units
16. Select *OK* twice

7.4.3 Running NSS

1. Switch to the *Hydrologic Modeling* module
2. Make sure Model combo box is set to *NSS*
3. Choose the *Select Basin* tool
4. Double-click on the icon for Basin 1B
5. Select *Yes*

Notice that the regression equation is automatically selected. Also, if our basin had overlapped with another NSS region, the areas and percentages of overlap for each region would also have been calculated.

6. Enter **10.8** for the Lake Area variable
7. Select the *Compute Results* button
8. Select *Done*

As you can see, the NSS Region coverage allows WMS to automatically load the appropriate regression equation(s) when we open the NSS dialog. However, this might not save us a great deal of time if we are only running the simulation once for a single basin. Nevertheless, if we plan to study many different basins on a regular basis, then creating an NSS Region coverage for our state would prove to be very efficient.

7.5 Conclusion

In this exercise, we have discussed the following concepts in conjunction with setting up an NSS simulation:

1. How to calculate important parameters with the Compute Basin Data command

2. How to use an NSS Region coverage to automatically determine which equations should be used and to compute any areas of NSS region overlap